Board Station for New Users Training Series

Module 2: Preparing a Design for PCB

Software Version 8.5_2

Mentor Graphics®

TABLE OF CONTENTS

TABLE OF CONTENTS [Continued]

TABLE OF CONTENTS [Continued]

Lesson 2

Editing a Schematic for PCB (Continued)

TABLE OF CONTENTS [Continued]

TABLE OF CONTENTS [Continued]

Lab 3

LIST OF FIGURES

LIST OF FIGURES [Continued]

LIST OF FIGURES [Continued]

LIST OF FIGURES [Continued]

LIST OF FIGURES [Continued]

LIST OF TABLES

About This Training

Welcome to the *Board Station for New Users Training Series*. For information on the tools you learn to use in this training series, see the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Workbook Organization

For an overview of the organization and content of all the modules of the *Board Station for New Users Training Series*, refer to section "Workshop Overview" in *Module 1: Introduction to Board Station.*

Related Documentation

For a complete listing of the manuals that make up the PCB documentation set, refer to section "Guide to the Documentation" in the *PCB Products Overview Manual*. The *PCB Products Overview Manual* describes how each manual can help you in the design process. You can find a listing of all Mentor Graphics manuals in the *Mentor Graphics Technical Publications Overview Manual*. Both these manuals are available in INFORM.

Documentation Conventions

For an explanation of the documentation conventions used in this workbook, refer to the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Installation Procedure

For complete instructions on installing the data for this module, refer to "Installation Procedure" in the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Lesson 1
Introduction to Design Architect with PCB Personality Module

This lesson introduces PCB features in Design Architect that help you prepare your schematic for Board Station.

Capture the Schematic and Simulate			Develop Libraries		
Schematic Editor	List Input	Simulators	Maps	Geometries	Part Numbers

Manage Changes — Changes / Back-Annotation

Package the Design	
Geoms Design Object	Packaging

Create the Design Layout	
Component Placement	Routing

Perform Thermal Analysis

Create Manufacturing Data			
Drawings	Artwork	Drill	Milling

Manufacture the Board

Figure 1-1. Board Process Flow Chart

Before you can design a board in Board Station, you need to provide a description of the circuit, either as a schematic created in Design Architect, or as ASCII files containing lists of net (wire) connections and components. Board Station tools accept either a Design Architect schematic or a nets file and a components file as design input.

If you are providing ASCII files of nets and components that describe your circuit, refer to *Board Station for New Users Training Series*, Module 4: *Packaging the Design for Layout*.

Objectives

This lesson discusses terminology and hierarchical design, and introduces you to the PCB Personality Module within Design Architect. In the lab you will view different areas of a schematic sheet and different hierarchical levels of a design.

After completing this lesson, you should be able to do the following:

● Explain what a hierarchical design is.

● Describe how to open any schematic sheet in Design Architect.

● View different areas of a sheet using strokes.

● Describe how to open down into the schematic of a symbol.

Terminology

Different fields within the electronics industry use different meanings for some terms. These terminology differences occur between front-end tools, such as Design Architect, and back-end tools, such as the PCB tools. The following list defines how some terms are used in Design Architect:

● **Component**: An object that contains and defines one or more models, such as schematics, VHDL descriptions, and symbols. A component also includes one or more interfaces, which are mechanisms that define which models are used at any one time.

- **Design viewpoint**: A configuration (set of rules) used by downstream applications to evaluate the design source object. A design can have one or more viewpoints for each application that reads the design. Design viewpoints are described in Lesson 3: "Creating Design Viewpoints" in this workbook.

- **Net**: A signal path, node, or wire that connects two or more pins. It is analogous to a wire or trace in a physical circuit.

- **Instance**: A unique version of a symbol model in the design database. A schematic might have several instances of one particular model, or one instance each of many models.

- **Model**: A representation of an aspect of a particular electronic circuit that tells a simulator how that circuit should behave. The two types of models and examples of each are:

  ○ **Functional models**—schematics, QuickParts, and VHDL models;

  ○ **Non-functional models**—symbol models, technology files, and library data technology files.

Location Map

The applications you will use require access to shared network resources such as component libraries. Mentor Graphics applications use a location map to locate the resources your workstation shares with others on the network.   A location map is an ASCII file that maps environment variables to hard pathnames that identify where the data is stored. You cannot use a soft pathname in a location map to specify the pathname of a variable.

Examples of variables whose pathnames are specified in location maps include $MGC_HOME, $MGC_GENLIB and $MGC_PCBPARTS. An example of a location map is shown in Figure 1-2.

```
$MGC_LOCATION_MAP_1

$MGC_HOME
/usr2/pad/v8.4_1.rls

$MGC_GENLIB
/net/wvfs/usr2/rls/hpu_v8.4/libs/gen_lib

$MGC_PCBPARTS
/net/wvfs/usr2/rls/hpu_v8.4/libs/pcb_parts
```

Figure 1-2. Location Map Example

Location maps are usually set up and maintained by a system administrator at each site. Contact your system administrator to learn how to access your location map and how to define the $MGC_LOCATION_MAP environment variable.

For information about location maps, refer to "Design Management with Location Maps" in the *Design Manager User's Manual*.

Design Hierarchy

A hardware design might require thousands of pieces of logic to accurately describe its overall function. One way to organize all of this information is through design hierarchy. A hierarchy consists of building one level of functional descriptions onto another. Each functional description progresses down the hierarchy, from a generalized description to a more detailed description.

To create a hierarchical design, you first create a schematic that has nothing but functional blocks connected with wires and buses, such as the example shown in Figure 1-3.
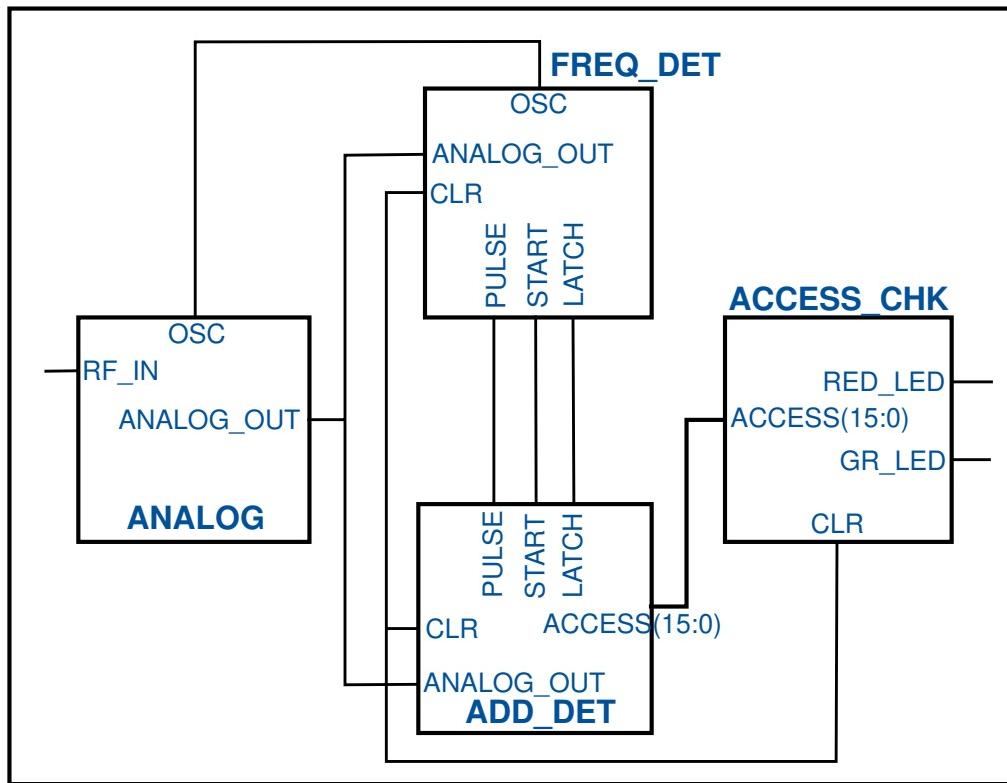
Figure 1-3. Functional Block Example

After you create the functional blocks, you can add the lower level descriptions to define each block. The description can have many forms, such as a schematic, VHDL, or PLA model. In Figure 1-4, Sheet1 describes the function of Component A.

True functional blocks are self-contained and work independently of each other. You can create them and test them separately. If changes are needed, you need only change the affected block of the design.
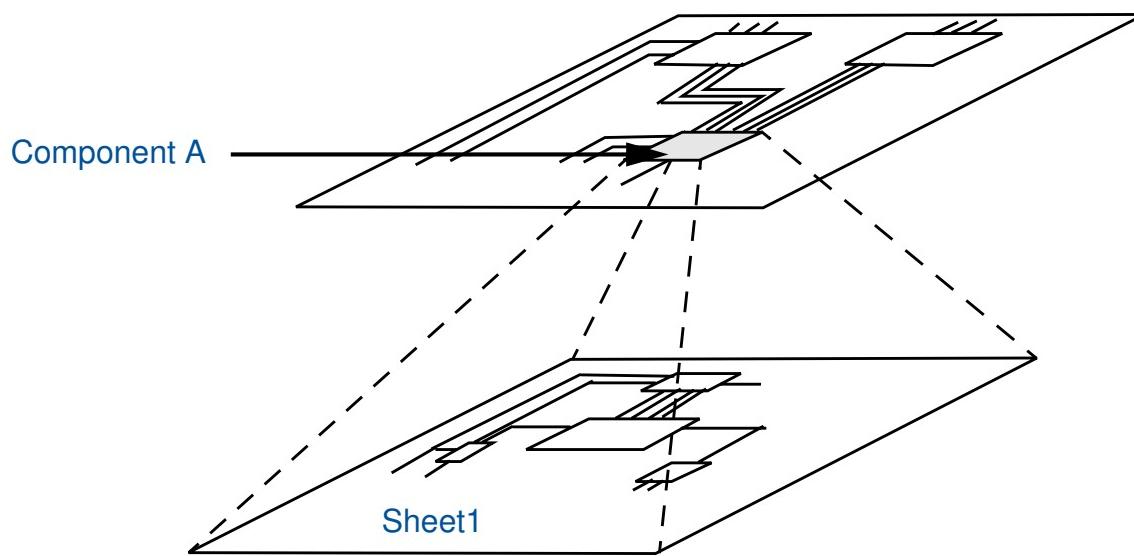
Figure 1-4. Design Hierarchy

Design Architect

Design Architect is the Mentor Graphics schematic capture tool. You invoke Design Architect from the Design Manager's Tool window by placing the cursor on the Design Architect icon and double-clicking the Select mouse button.

Design Architect includes a Schematic Editor, a Symbol Editor, a VHDL Editor, and an optional Logical Cable Editor. You use these editors to create and edit logical designs for PCB layout, IC layout, and analog and digital simulation.

You invoke Design Architect editors by choosing a popup menu item, clicking on the appropriate palette icon, or pressing the desired function key. The Session popup menu and palette menu are illustrated in Figure 1-5.
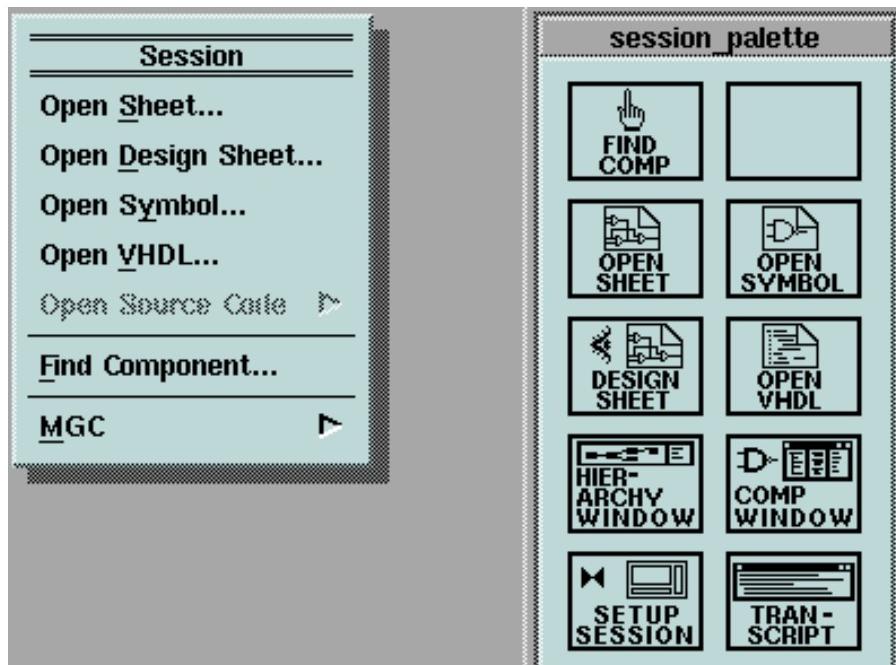
Figure 1-5. Session Popup Menu and Palette Menu

When you invoke an editor, a new window is displayed within the
Design Architect Session window; the menu bar, palette menu, and
soft keys are replaced by the menus and soft keys for that editor.
Figure 1-6 shows the Design Architect window with two schematic
windows open. Different areas of the window are also identified in the
illustration.

The Symbol Editor is described on page 1-9, and the Schematic Editor
is discussed on page 1-32.

For information about creating VHDL models, refer to the
System-1076 Design and Model Development Manual.

The Logical Cable Editor is a schematic editor with additional features
for adding cables and connectors. After you create a Logical Cable
schematic, you either import the design into the Physical Cable
application, or use it to create a wire list containing all the connectivity
information of the schematic. For information about the Logical Cable
Editor, refer to the *Logical Cable User's Manual* and the *Logical
Cable Reference Manual*.

Window
menus

Menu bar

Active window

Minimize

Maximize

Status line

Window banners

Palette menu
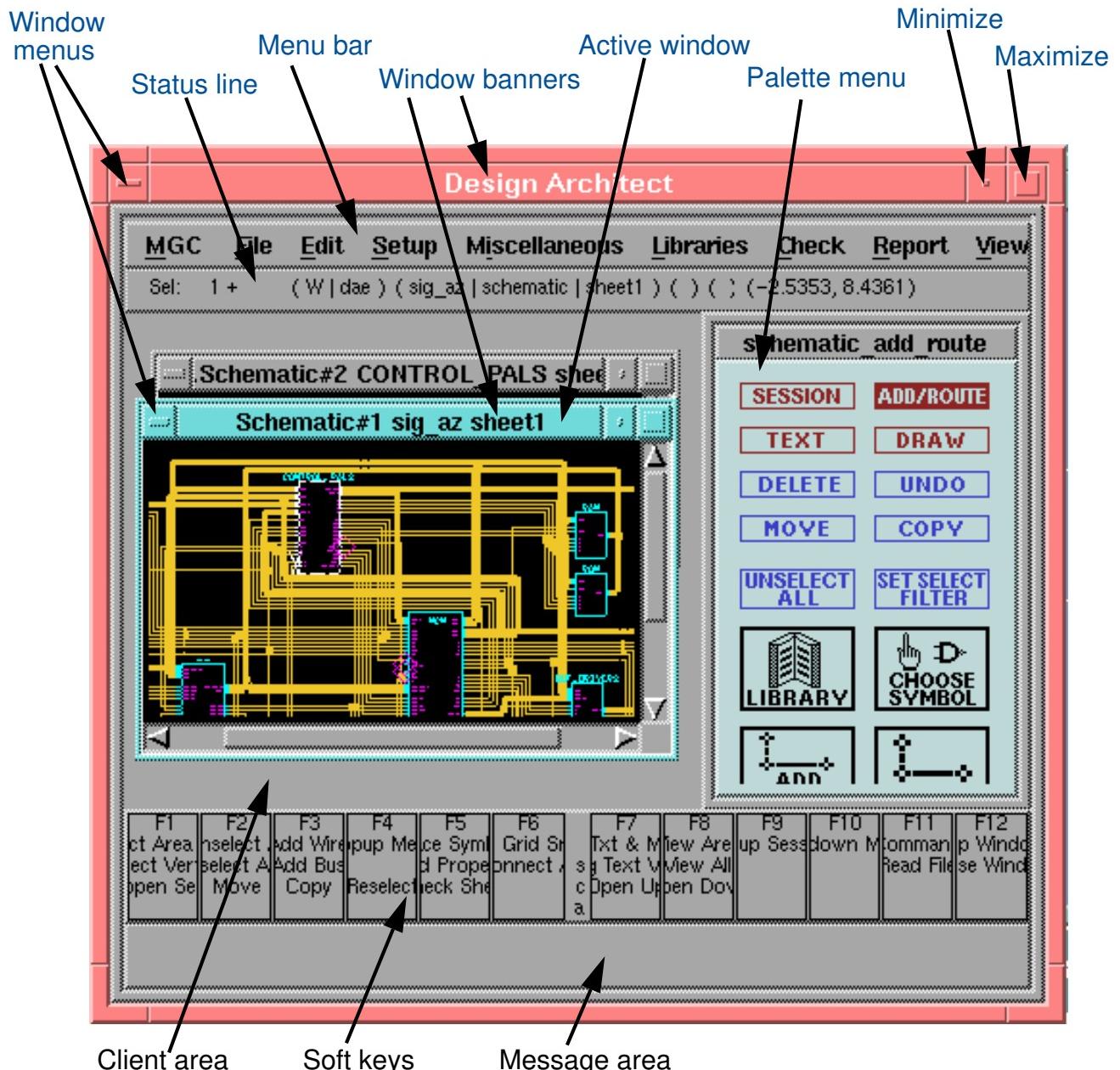
Figure 1-6. Design Architect Window

Client area   Soft keys   Message area

Symbol Editor

You use the Symbol Editor to create and edit component symbols to place on your schematic sheets. A component symbol can represent anything, from a discrete component that you might buy off the shelf, to a complete system design. The functionality of the circuit represented by a symbol does not need to be defined when you create the symbol.

Symbols for many components are available in Mentor Graphics component libraries. Mentor Graphics component libraries are discussed on page 2-3 of this workbook.

To invoke the Symbol Editor from the Design Architect Session window, click on the **Palette > Open Symbol** icon. This displays the Open Symbol dialog box, shown in Figure 1-7.
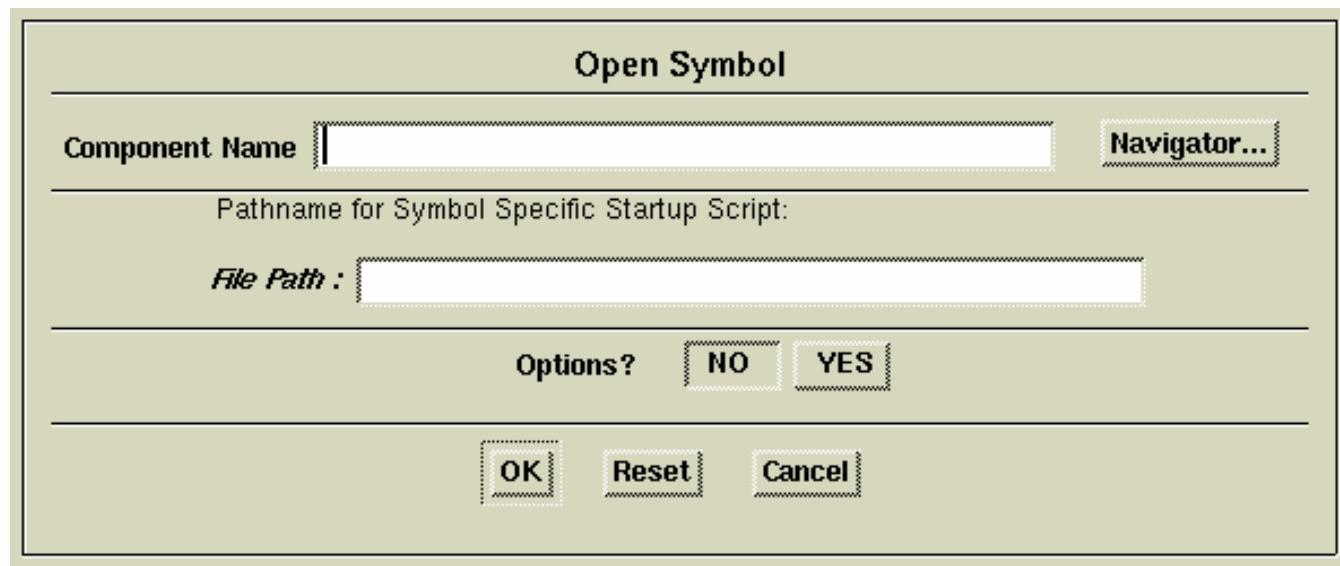
Figure 1-7. Open Symbol Dialog Box

Enter the component pathname for the symbol, or click the Navigator button to display the Dialog Navigator shown in Figure 1-8.
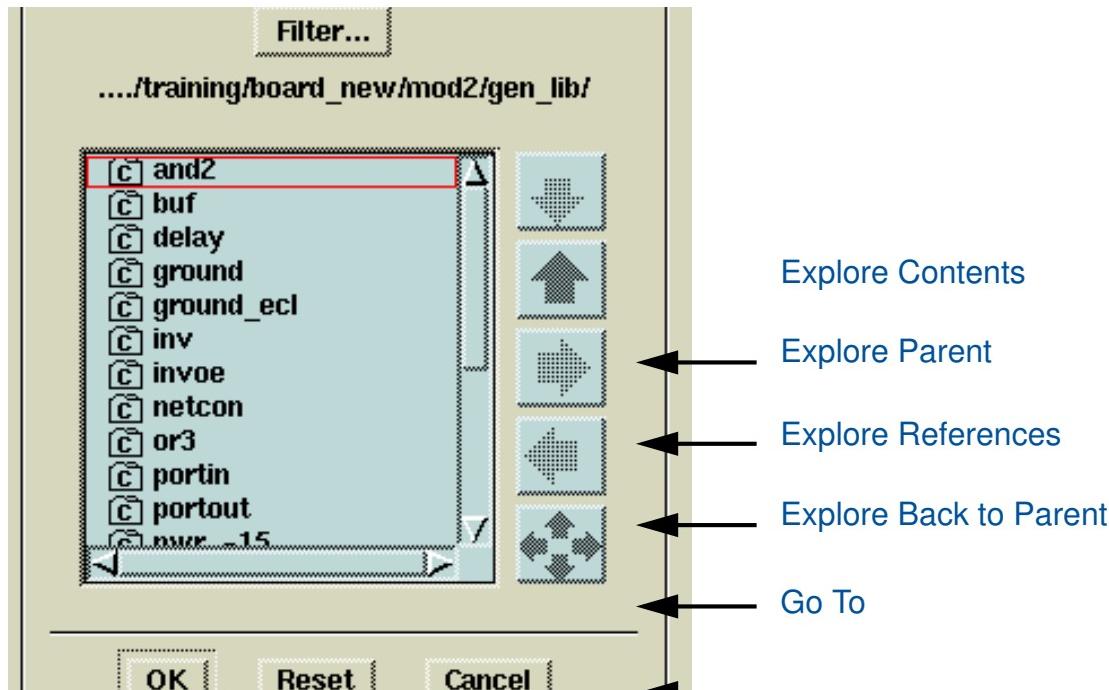
Figure 1-8. Dialog Navigator

Most dialog boxes requiring a pathname have a Dialog Navigator to help you find design objects. The buttons on the right side of the Navigator allow you to navigate through the file system, or to navigate references, as follows:

- **Explore Contents**—displays design objects contained in the selected directory.

- **Explore Parent**—displays the parent directory of the selected design object.

- **Explore References**—replaces the currently displayed list with the references of the selected design object.

- **Explore Back to Parent**—navigates back to the design object that references the currently displayed objects. This button is only activated when you explore the reference of an object.

- **Go To**—displays a dialog box in which you enter the pathname of the directory to which you want to navigate.

Choose the desired component in the Dialog Navigator list by clicking the Select mouse button on the symbol name, and clicking the **OK** button on the Navigator. The pathname is automatically placed in the text entry box.

When you click the **OK** button on the Open Symbol dialog box, Design Architect opens a Symbol Editor window in which the symbol you specified is displayed. If the symbol does not exist, an empty sheet is displayed for you to create a symbol. The various menus contain all the functions you need to create a new symbol or modify an existing symbol. The Symbol Editor palette menus are shown in Figure 1-9.
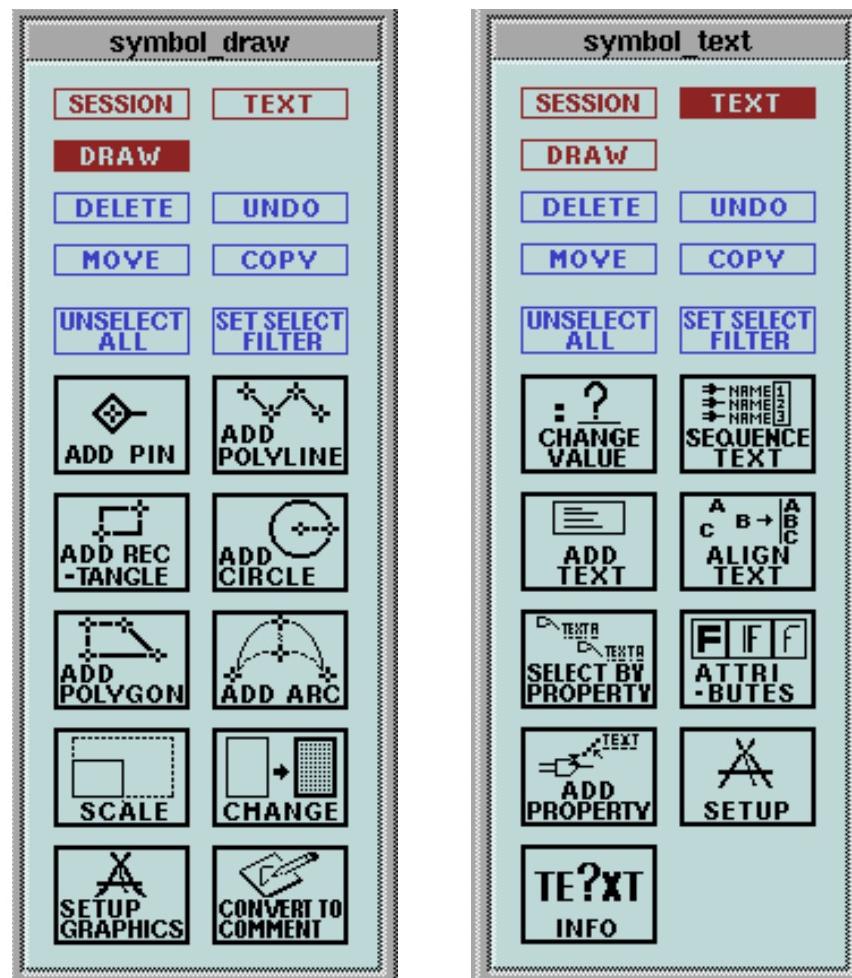
Figure 1-9. Symbol Editor Palette Menus

Popup menus in the Symbol Editor are *selection sensitive* by default, which means the selected objects determine which popup menu is displayed. Each popup menu includes the menu items you are most likely to need for the selected objects. For example, if you select symbol pins, the **Symbol Body & Pins** menu includes functions for adding properties, and moving, copying, and aligning the selected objects.

If nothing is selected, the **Add** menu contains all the functions you need to create a symbol. The first item in each menu, *Other Menus*, lets you move to any other popup menu. There is also a *selection free* popup menu which can be displayed regardless of what objects are selected. Figure 1-10 shows the Symbol Editor selection-sensitive popup menus.
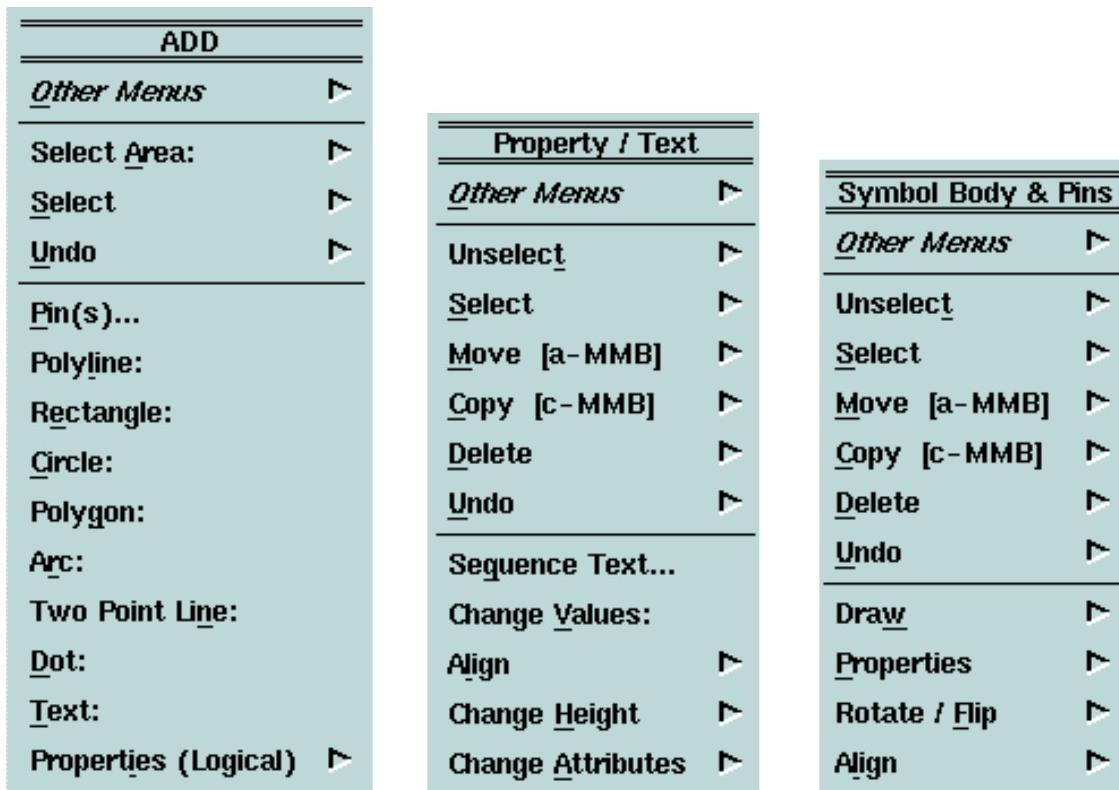
Figure 1-10. Symbol Editor Selection Sensitive Popup Menus

Creating a Symbol

A symbol is a representation of a functional model, such as a simple logic function, a complex IC, or even a complete design. A component symbol has four basic parts:

- **Symbol body (shape).** The graphical image of the symbol.

- **Pins.** Connection points when an instance of the symbol is placed on a schematic sheet.

- **Origin Point.** The reference point used to place the symbol on a sheet.

- **Properties.** Provide information about the function of the symbol. Properties are discussed in Lesson 2: "Properties" on page 2-12.

The Symbol Editor window contains two grids, shown in Figure 1-11, to help you locate and position objects and text. The course grid with the heavier marks is the Pin Grid, which constrains electrical objects. Symbol pins must fall in this grid. The finer grid is the Snap Grid, which controls placement of non-electrical objects. Symbol body graphics can be snapped to this grid.

Snap Grid   **Pin Grid**

Symbol#3 74ls244a
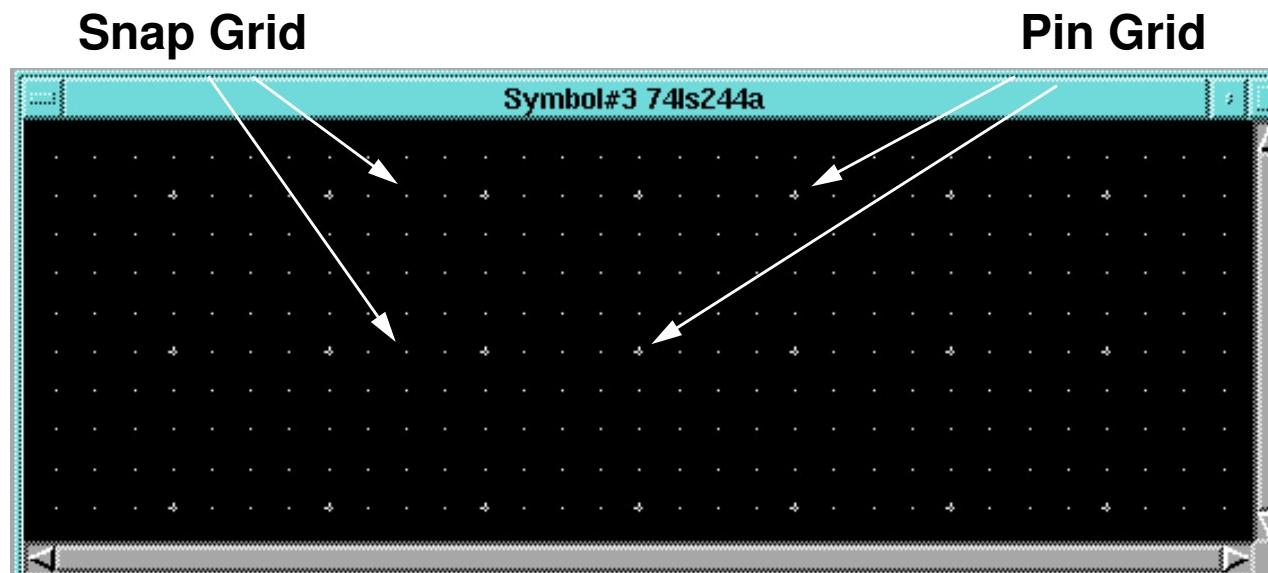
Figure 1-11. Symbol Grid

Symbol Body

You can draw a rectangular body, or any shape consisting of line segments, arcs, and circles. Table 1-1 shows the number of location points and the mouse actions required.

Table 1-1. Symbol Graphics

Graphics	Prompts	Action
Polyline	initial, end segments (2 minimum)	Click at initial point; click at end of each segment. Double-click to terminate.
Rectangle	diagonal corners (2)	Press at one corner, drag, release to define opposite corner of rectangle.
Arc	initial, end, arc (3)	Click at each end, then at arc point.
Polygon	polygon points (3 minimum)	Click at each point of polygon, double-click to end; segment from last to first point is automatically drawn.
Circle	center, radius (2)	Press button at center, drag, release at radius point.
Two Point Line	initial, end (2)	Press button at one end, drag to other end, release.
Dot	point (1)	Click at dot location.
Text	text, location point (1)	Enter text in prompt bar, drag image of text, and click at desired text location.

To draw a symbol body, click on the appropriate palette icon, such as **Add Rectangle**, or choose one of the items from the **Add** popup menu, and follow the actions described in Table 1-1.

Symbol Pins

You can create pins either before or after you create the symbol body. To add pins, click on the **Symbol_Draw Palette > Add Pin** icon. This displays the Add Pin(s) dialog box shown in Figure 1-12.
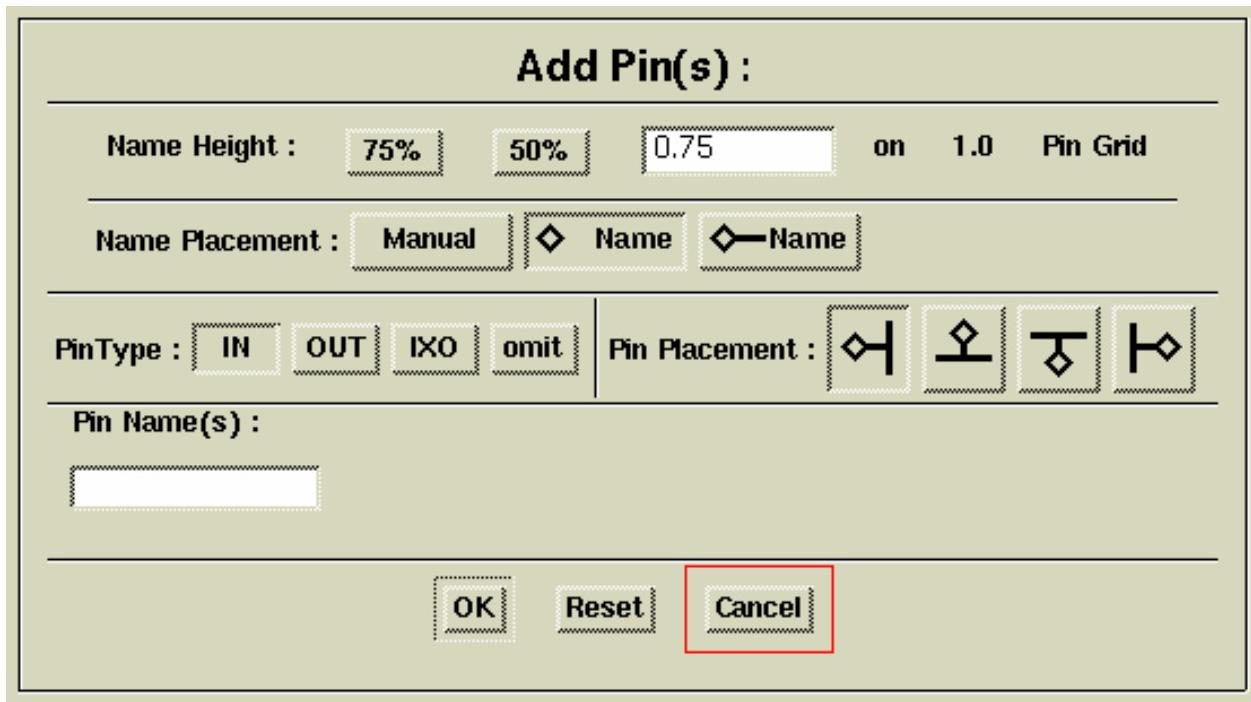
Figure 1-12. Add Pin(s) Dialog Box

In the Add Pin(s) dialog box:

- **Name Height** indicates the height of the pin name with respect to the pin grid.

- **Name Placement** has three types of pin name placement:

  ○ **Manual**. You position the pin name and text location manually.

  ○ **Name (with diamond)**. The graphical pin indicator is created and the pin name is positioned next to them.

❍ **Name (with diamond and whisker)**. The graphical pin indicator and whisker are created and the pin name is positioned next to it. Whiskers are short lines that project from the border of the symbol body to indicate where input and output pins are connected.

● **Pin Type** is a property associated with each pin, and can be either IN, OUT, IXO, or omitted.

● **Pin Placement** designates if the pin is placed to the left, top, bottom, or to the right of the symbol body.

● **Pin Name** (Pin property) is the logical pin name, such as *in_1*, *D*, or *CLK*.

After you complete and **OK** the dialog box, a prompt bar appears with the information for the first pin, and prompts you for the location to place the pin. When you click the Select mouse button at the desired pin location, the pin, whisker (if specified), and text are placed. The prompt bar repeats for each pin name you specified in the dialog box.

After creating the symbol body and adding pins, you add connectivity information and design characteristics in the form of properties, which are discussed in Lesson 2: "Properties" on page 2-12.

Checking and Saving the Symbol

A symbol must pass certain checks before you can use it on a schematic sheet. The following check categories are required by Mentor Graphics applications:

● **Symbol pin**. Verifies that at least one pin is present on the symbol, all symbol pins have unique names, and symbol property values have valid expression syntax.

● **Symbol body**. Verifies the existence of symbol graphics, checks expression syntax in property values, and looks for duplicate property names on the symbol body objects.

- **Special symbol**. Verifies proper construction of a symbol that represents a special instance, such as a bus ripper or on/offpage connectors.

To check using the default check settings, choose the **Check > With Defaults** pulldown menu item. Checking results are displayed in a Check Report window, shown in Figure 1-13.
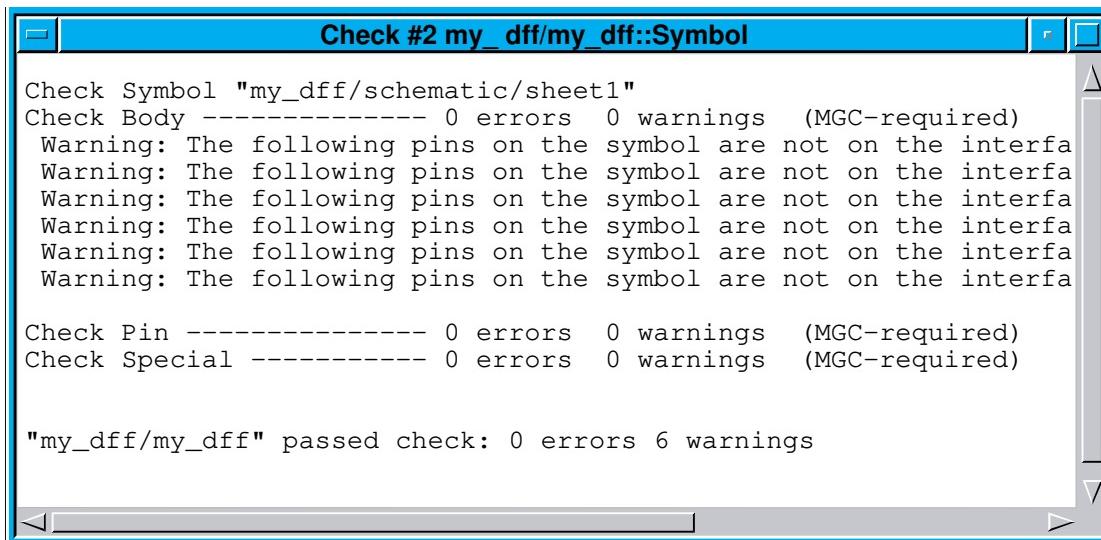
```
Check #2 my_ dff/my_dff::Symbol

Check Symbol "my_dff/schematic/sheet1"
Check Body ------------- 0 errors  0 warnings  (MGC-required)
 Warning: The following pins on the symbol are not on the interfa
 Warning: The following pins on the symbol are not on the interfa
 Warning: The following pins on the symbol are not on the interfa
 Warning: The following pins on the symbol are not on the interfa
 Warning: The following pins on the symbol are not on the interfa
 Warning: The following pins on the symbol are not on the interfa

Check Pin --------------- 0 errors  0 warnings  (MGC-required)
Check Special ----------- 0 errors  0 warnings  (MGC-required)


"my_dff/my_dff" passed check: 0 errors 6 warnings
```

Figure 1-13. Check Report Window

Always check your symbol before you save it. If your symbol is not successfully checked, it can still be saved to disk, but it cannot be instantiated. To save your symbol, choose the **File > Save Symbol** menu item.

For complete instructions on creating and modifying symbols, refer to *Getting Started with Design Architect*, *V8 Design Architect Training Workbook*, and in "Procedures" in the *Design Architect User's Manual*.

The Schematic Editor

You use the Schematic Editor to create and modify schematic diagrams. You can place component symbols on a schematic sheet, add the wiring, and assign properties to supply design information. Properties are discussed beginning on page 2-12 of this workbook.

To open a Schematic Editor window from the Design Architect Session window, click the Select mouse button on the **Palette > Open Sheet** icon. The Open Sheet dialog box prompts you for a component name. This dialog box also has the Navigator to help you choose the component name. If the component you specify does not exist, a new component is created with the name you supply.

Open Sheet

Component Name /user/jeanf/training/board84nwp/mod2/sig_az   Navigator...

Sheet : sheet1   *Options...*

Startup File Path:

Open as :

◆   ◇   ◇
Editable   Read Only   Show Hierarchy

OK   Reset   Cancel
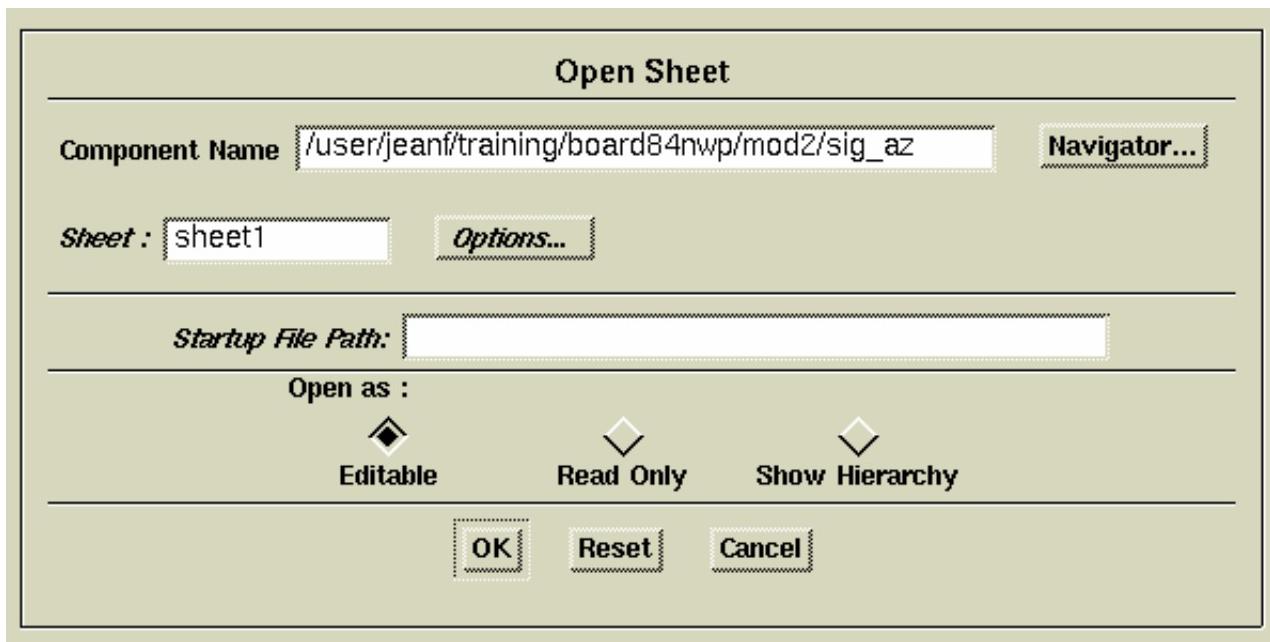
Figure 1-14. Open Sheet Dialog Box

A component is any design created by Design Architect. A component can be a design of a single part, or it can be a design of a complex, multi-sheet hierarchical system.

The Schematic Editor has three palette menus, shown in Figure 1-15. **Add_Route** is the default palette menu, and includes functionality for adding symbols and wires to a schematic sheet. The **Text** palette lets you add or modify text and text attributes on a schematic.

The **Draw** palette includes the functions you need for drawing a symbol in place on a schematic sheet, and for drawing comment objects, such as title blocks and sheet borders.
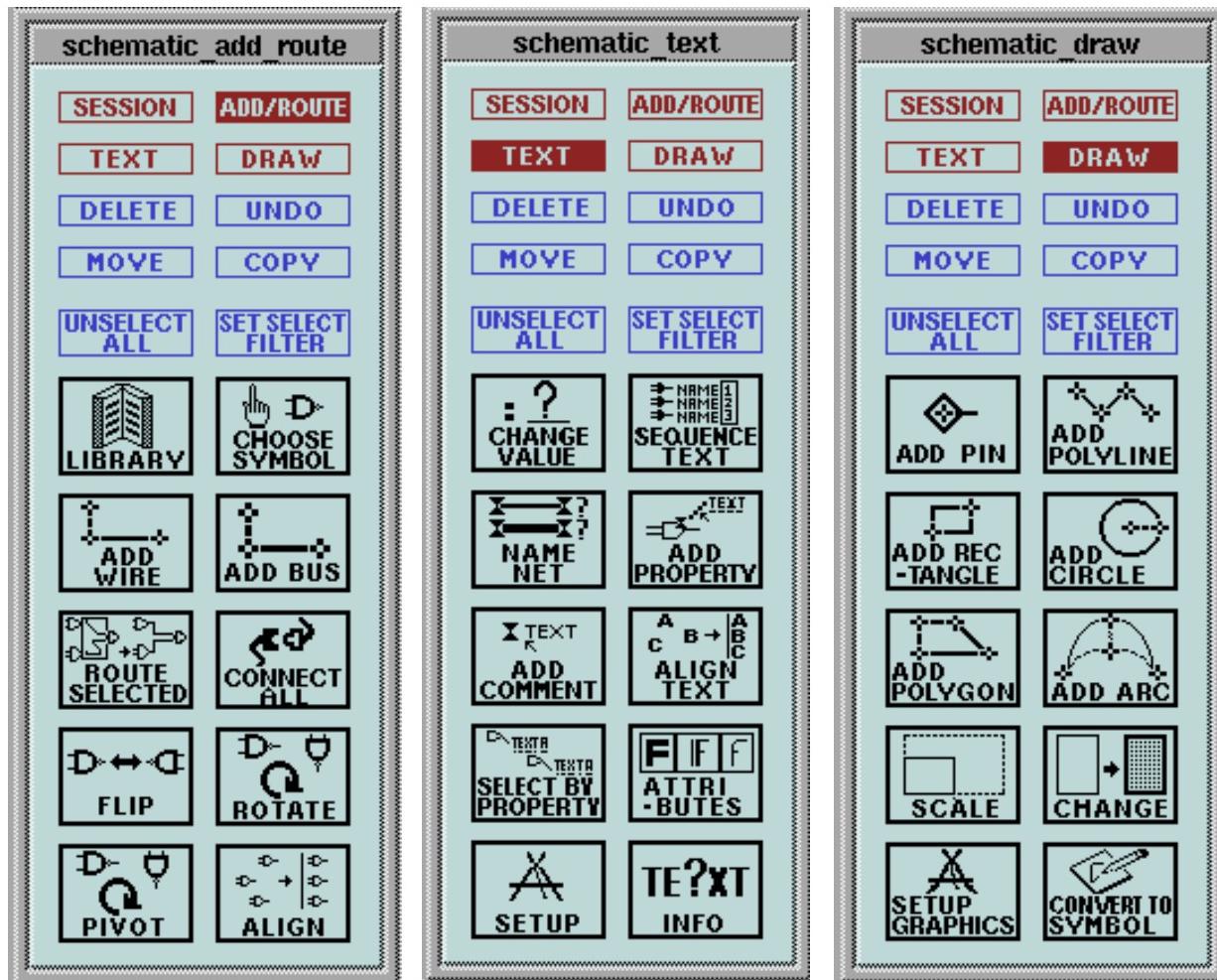
Figure 1-15. Schematic Editor Palette Menus

Popup menus in the Schematic Editor are also selection-sensitive; the selected objects determine which popup menu is displayed. The first item in each menu, *Other Menus*, lets you move to any other menu. A selection-free popup menu can be displayed regardless of what objects are selected. The Schematic Editor selection-sensitive popup menus are shown in Figure 1-16.

You learn about creating a schematic in Lesson 2: "Editing a Schematic for PCB" on page 2-1.

ADD	
Other Menus	▷
Select Area	▷
Select	▷
Undo	▷
Wire:	
Bus:	
Instance	▷
Frame:	
Draw	▷
Connections	▷

Mixed Selection	
Other Menus	▷
Unselect	▷
Select	▷
Move [a–MMB]	▷
Copy [c–MMB]	▷
Delete	▷
Undo	▷
Change Net	▷
Connections	▷
Route	▷
Rotate / Flip	▷

Property / Text	
Other Menus	▷
Unselect	▷
Select	▷
Move [a–MMB]	▷
Copy [c–MMB]	▷
Delete	▷
Undo	▷
Sequence Text...	
Change Values...	
Align	▷
Change Height	▷
Change Attributes	▷

NET	
Other Menus	▷
Unselect	▷
Select	▷
Move [a–MMB]	▷
Copy [c–MMB]	▷
Delete	▷
Undo	▷
Name Nets:	
Properties	▷
Add Wire:	
Add Bus:	
Change Net	▷
Connections	▷
Route	▷

Instance	
Other Menus	▷
Unselect	▷
Select	▷
Move [a–MMB]	▷
Copy [c–MMB]	▷
Delete	▷
Undo	▷
Properties	▷
Update	▷
Replace	▷
Rotate / Flip	▷
Align	▷
Open Down...	

Draw	
Other Menus	▷
Unselect	▷
Select	▷
Move [a–MMB]	▷
Copy [c–MMB]	▷
Delete	▷
Undo	▷
Properties	▷
Scale:	
Add	▷
Change	▷
Convert to Symbol...	

Figure 1-16. Schematic Editor Selection Sensitive Popup Menus

Opening Down into a Component

When you specify a component name in the Open Sheet dialog box, by default you are opening sheet1 of the top level of the design. You can open any sheet of any level of hierarchy by using the navigator to find the particular sheet you want, or by specifying a sheet name.

If you have already opened a sheet, you can open down into a selected component on that sheet. Figure 1-17 shows the simple representation of design hierarchy that was discussed earlier in this lesson.
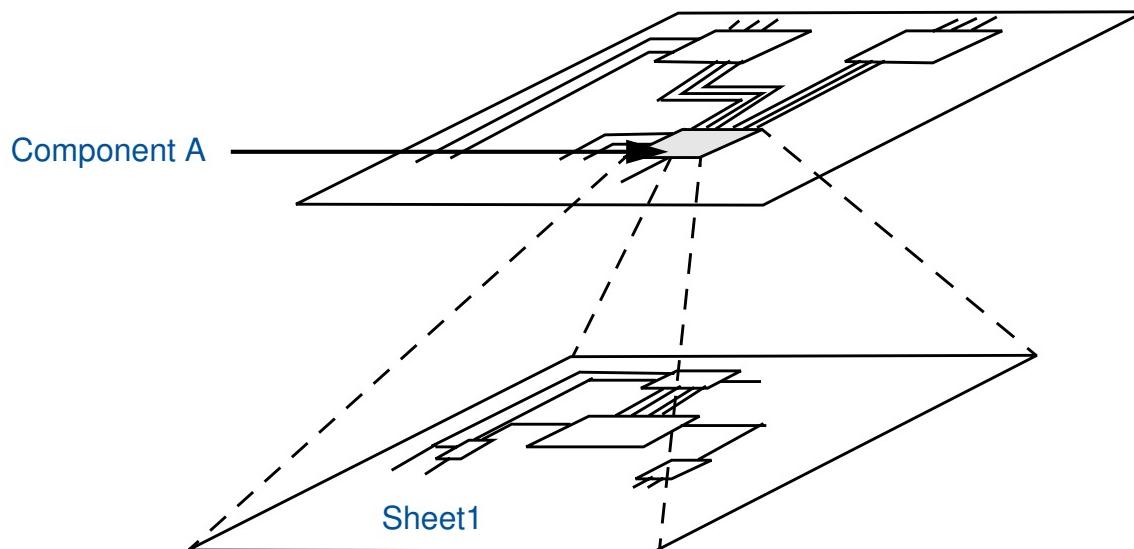
Component A

Sheet1

Figure 1-17. Opening Down into a Component

If you are viewing the top level sheet in Figure 1-17 and you want to open the sheet that represents Component A, you would perform the following steps:

1. Press the Unselect All function key to be sure nothing is selected.

2. Click the Select mouse button on Component A.

3. Choose the **File > Open Down > Choose Model** pulldown menu item, as shown in Figure 1-18.
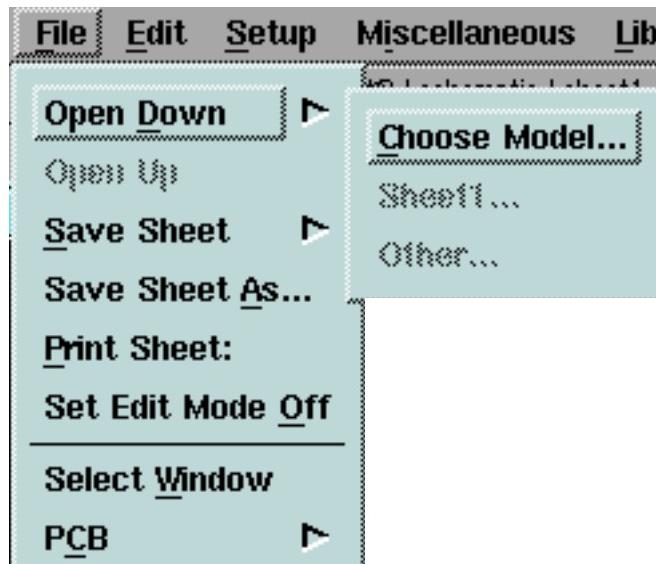
Figure 1-18. File > Open Down Menu

A list of available models is displayed in the Open Down dialog box, shown in Figure 1-19.
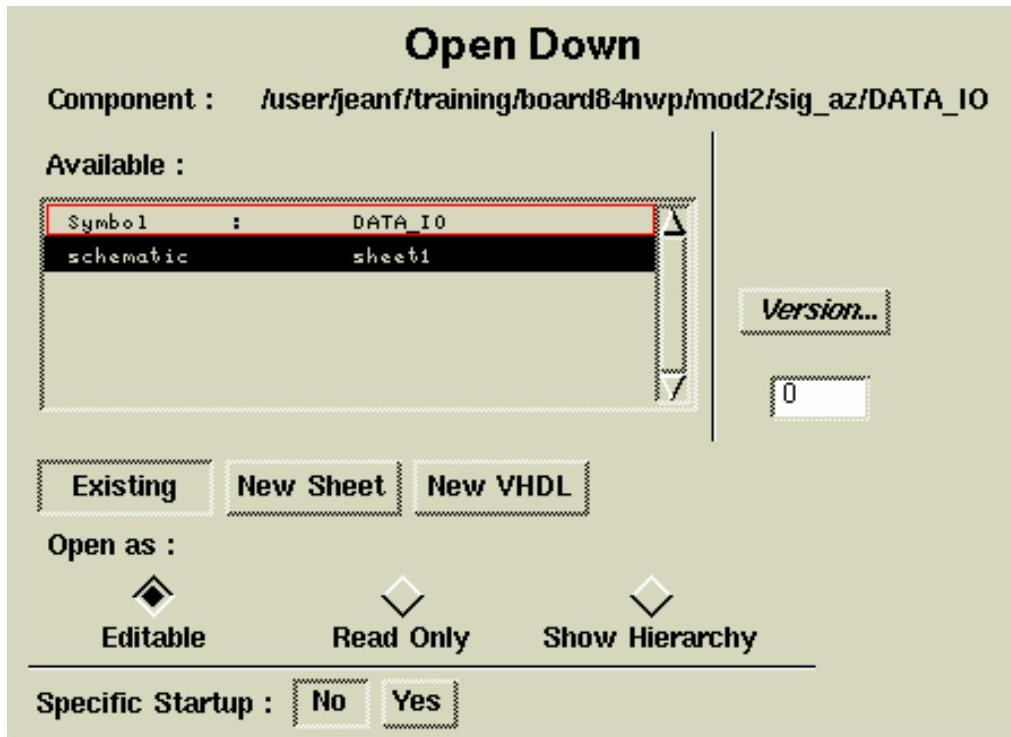
Figure 1-19. Open Down Dialog Box

4. Click the Select mouse button on *schematic* to open Sheet1 of Component A in a new Schematic window.

The model you choose determines which type of Edit window is opened. If you choose *symbol*, the Symbol Editor is opened. If you choose a VHDL model, the VHDL Editor is opened.

To return to the upper-level sheet, pop the window displaying the upper-level sheet to the top, or close the lower-level sheet.

PCB Personality Module

The PCB/DA Personality Module is an optional software package provided with Board Station. The module adds functionality to Design Architect to facilitate adding PCB properties and specifying design rules.

If the PCB/DA Personality Module is loaded on your software tree, it will *announce* itself when you invoke Design Architect. The first line displayed in the shell after invocation (whether invoked from Design Manager, or from a shell) will include the following:

```
//  pcb_da (PCB/DA Personality Module)
```

The invocation line will also show you the PCB Personality Module version number and the date it was built, and the Design Architect version number and the date it was built.

A personality module is customized userware that is loaded on top of Design Architect. The userware in the PCB/DA Personality Module includes menu items to give the electrical engineer some control over layout issues, such as grouping components on the board and specifying package types. Rules and requirements specified by the electrical engineer are automatically fed into downstream processes, such as PACKAGE and LAYOUT, as needed.

Setup > PCB

You can use the **Setup > PCB** submenu items, shown in Figure 1-20, to define the physical layer configuration and net design rules.
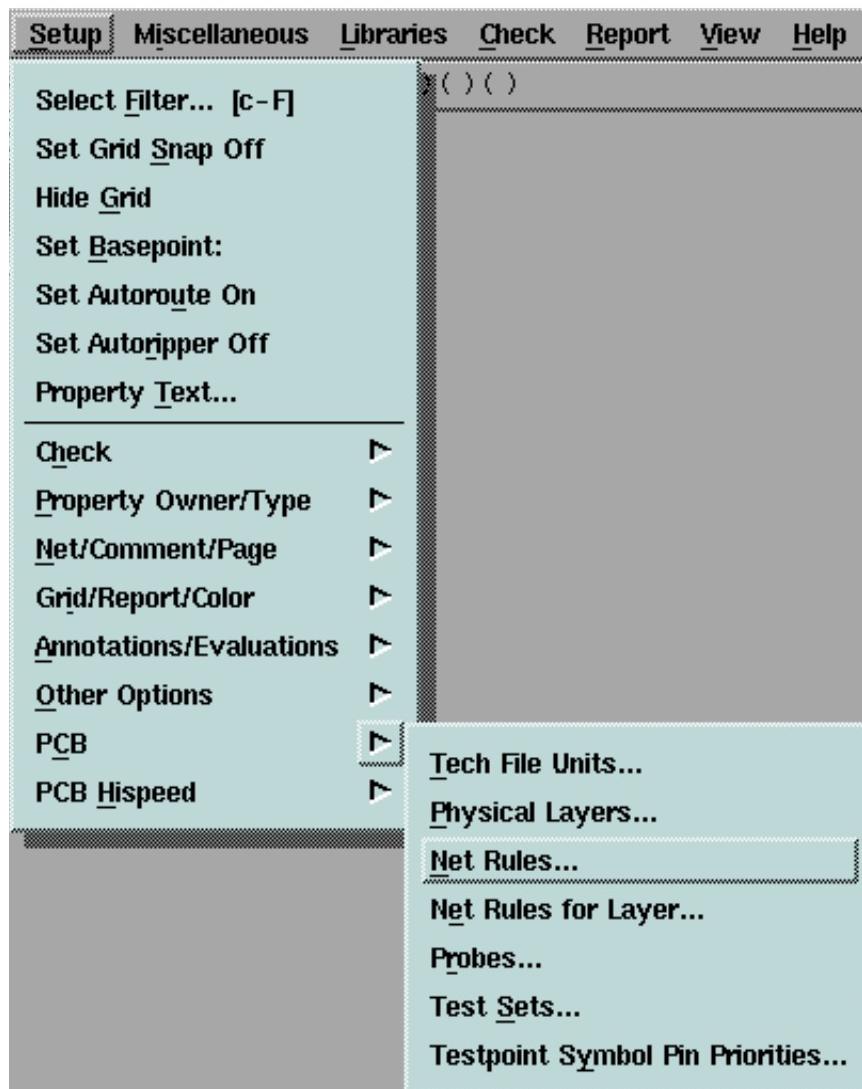
Figure 1-20. Setup > PCB Submenu

If you have the optional Testpoint application, you can also use the last three submenu items to set up test probes, test sets, and testpoint symbol pin priorities.

The **Setup > PCB Hispeed** submenu items let you define electrical class rules, board materials, layer materials, and conversion of timing delays into physical length constraints.

When you set up design rules, you need to set units of measurement for the board first. To do this, choose the **Setup > PCB > Tech File Units** menu item. The dialog box shown in Figure 1-21 lets you specify units for the board. If you choose one of the other PCB submenu items before setting the units, the system will remind you to set the units first, and display this dialog box.
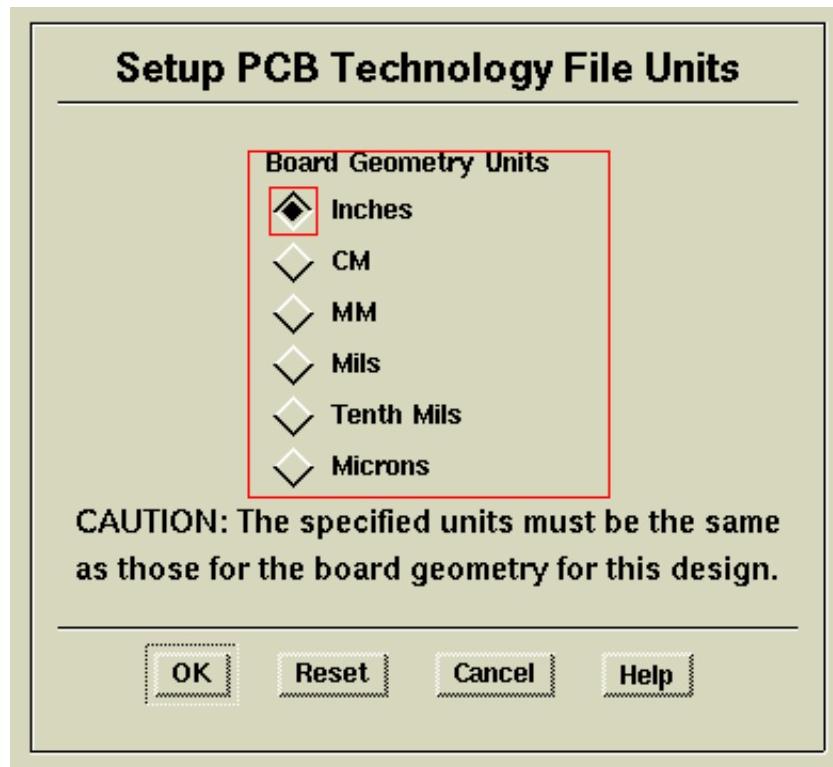
Figure 1-21. Setup PCB Technology File Units Dialog Box

After you specify the units of measurement, you can set up design rules for specified nets or all nets, and for nets on specified layers or all layers. You set up these rules by choosing the **Setup > PCB > Net Rules** and **Setup > PCB > Net Rules for Layer**, respectively.

You specify design rules by adding property name/value pairs to design objects. The PCB Personality Module defines **PCB Properties** submenus which you can access from most popup menus and from the **Edit > Edit Commands** pulldown menu. Figure 1-22 shows the **PCB Component Properties** submenu, and Figure 1-23 shows the **PCB Pin Properties** submenu.

Properties are discussed in "Properties" on page 2-12 of this module. Design rules are discussed in detail in *Board Station for New Users Training Series*, Module 7: *Routing Traces on a Circuit Board*.
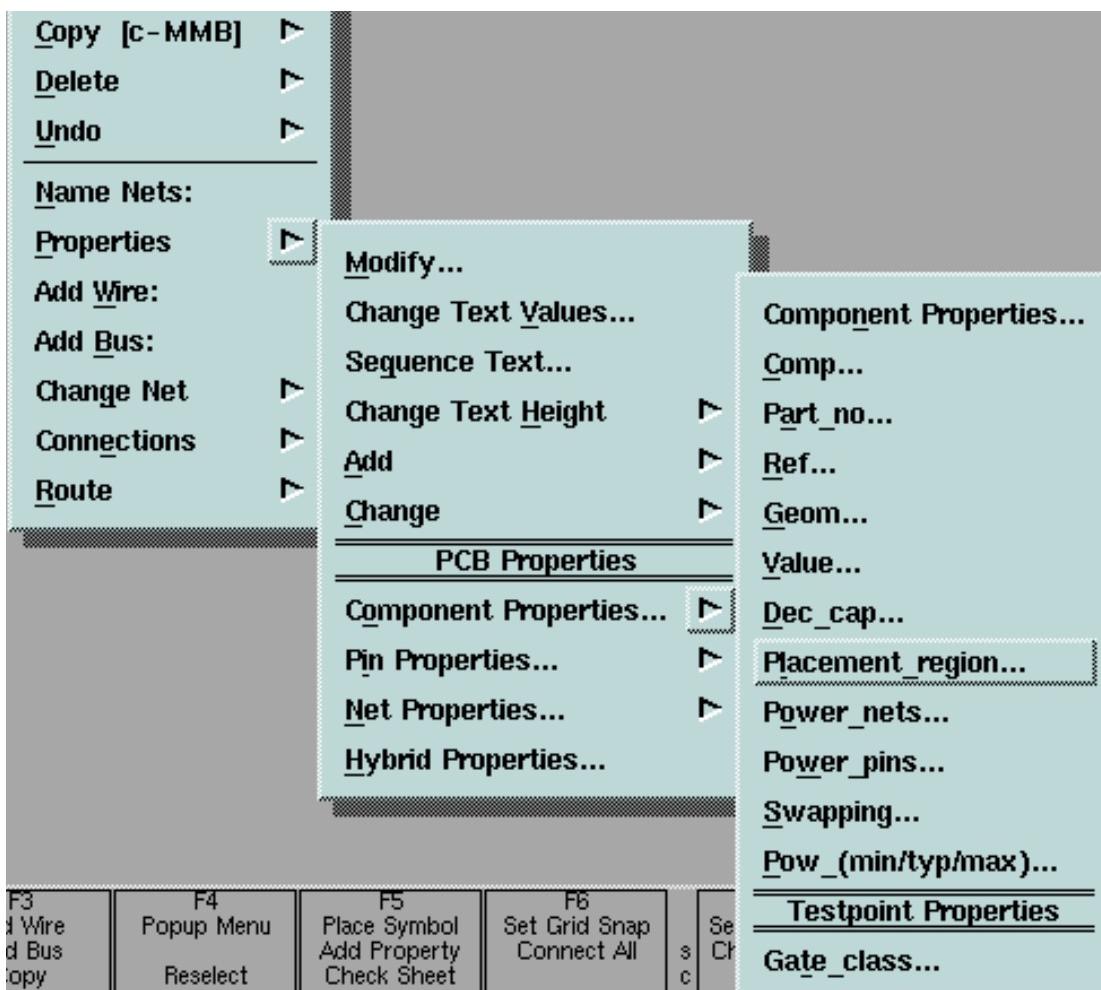
Figure 1-22. PCB Component Properties Submenu
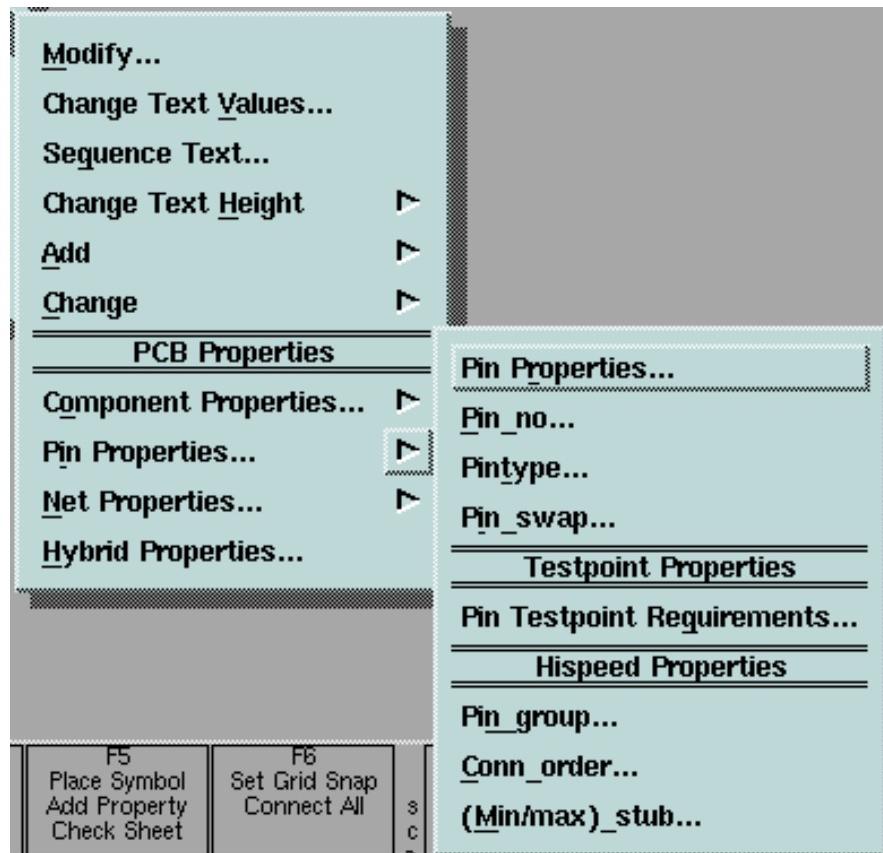
Figure 1-23. PCB Pin Properties Submenu

File > PCB

If you specify design rules (either new or modified) in Design Architect, you need to save them using the **File > PCB > Save Technology** menu item, shown in Figure 1-24.

The **Save Technology** menu item saves design rules in a special *pcb* container in your design. If a *pcb* container does not exist in your design, the system creates one automatically. Be sure to save a new or modified technology design object before exiting Design Architect, or those design rules will not be available to the PCB tools.
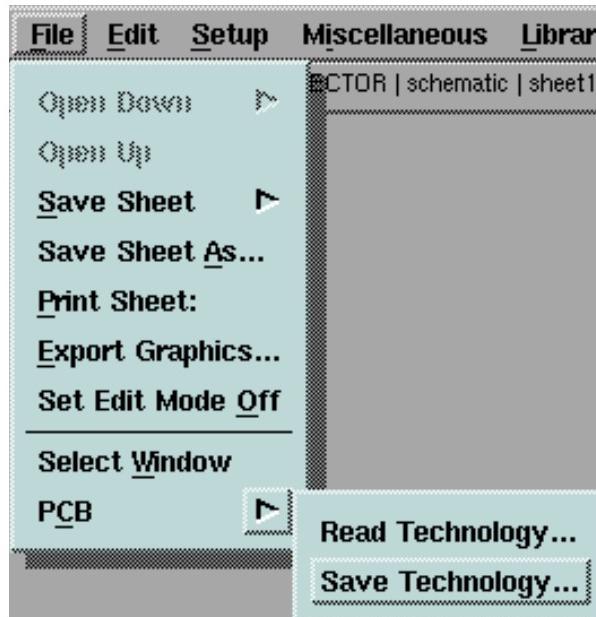
Figure 1-24. File > PCB Submenu

To read existing design rules into Design Architect, choose the **File > PCB > Read Technology** menu item, and when the dialog box is displayed, specify the name of the desired technology file in the text entry field.

Selection Concepts

You select objects so that you can act upon them with menu items such as Move, Copy, or Delete. You can select a single object by clicking the Select mouse button on that object. This is called *point selection*. If you click the Select mouse button on a selected object, it becomes unselected.

You can select several objects by defining a selection rectangle around them. Do this by pressing the Select mouse button at one corner of the rectangle, moving the cursor to the diagonally opposite corner and releasing the mouse button. This is called *area selection*.

Selection Sets

As you select objects or create objects, they are added to a selection set. A selection set remains open until you perform an edit or report operation on the objects. The edit or report operation closes the selection set, meaning no more objects are added to the set, although the objects remain selected. The next Select command opens a new selection set and unselects all of the previously selected items.
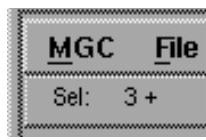
The Select Count in the status line shows whether the selection set is open or closed. If the number of selected items is followed by a plus (+) character, as shown at the left, the selection set is open; otherwise, it is closed (*Sel: 3*).

Each open sheet or symbol has its own selection set. If multiple windows are open on the same sheet, selected objects are highlighted in each of those window.

Selection Menus

You can use the Select menu items to choose the types of objects you want to select. For example, you can select all instances on a schematic sheet by choosing **Select > All > Instances** from a popup menu. You can select all pins in an area by choosing **Select > Area > Pins** and defining the selection area with the mouse.

All of the popup menus include selection items. If you choose any of the **Select** submenu items other than **Filtered** or **Anything**, the selection overrides the select filter (discussed in the next subsection) for that one time only.

When you choose any of the **Select > Area** menu items, a prompt bar displays, prompting you for a selection rectangle. You can click on the **Options** button on the prompt bar to display a dialog box in which you can choose more than one type of selection. Figure 1-25 shows the **Select > Area** submenu. Figure 1-27 shows the Select Area Switch Options dialog box.
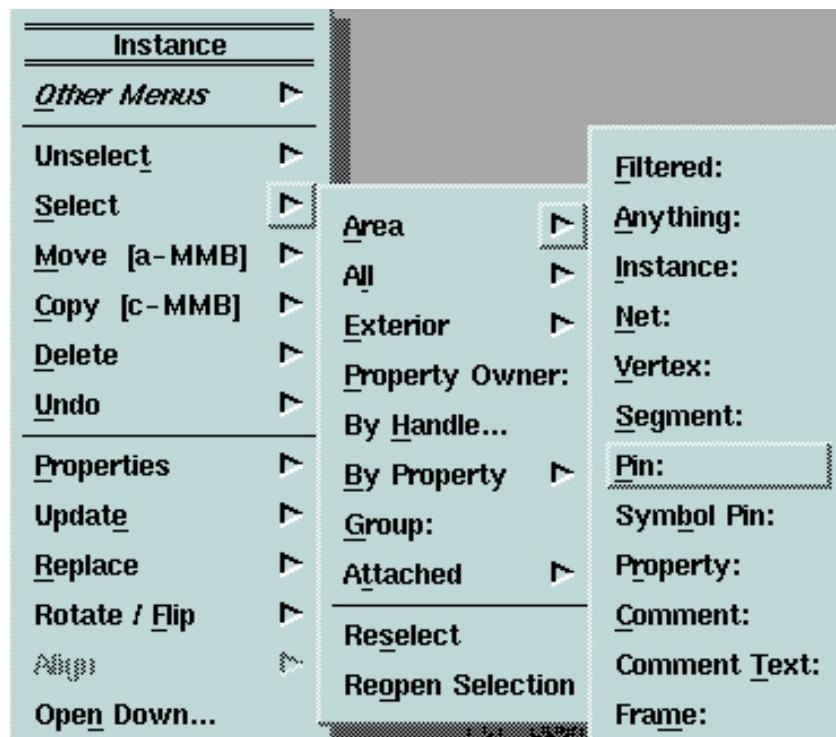
Figure 1-25. Select > Area Submenu and Prompt Bar
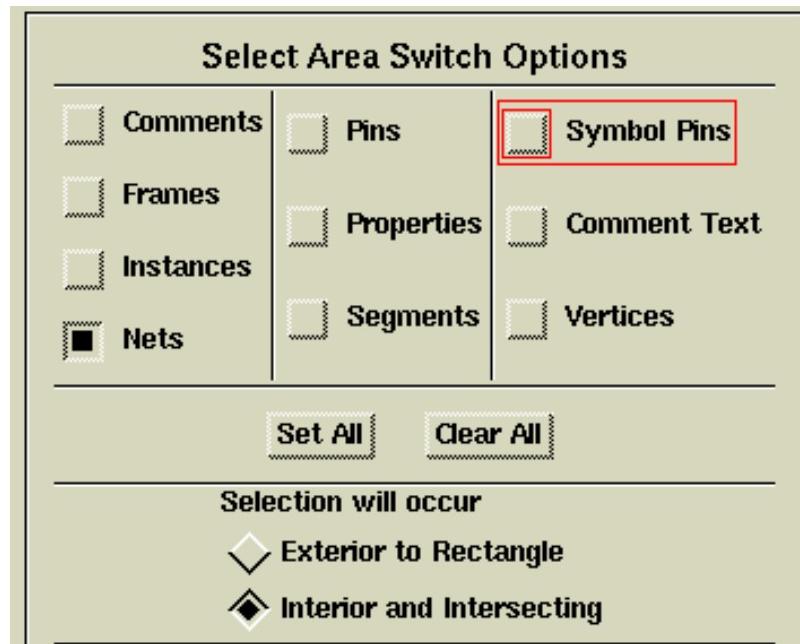
Figure 1-26. Select Area Switch Options Dialog Box

Select Filter

SET SELECT FILTER

The select filter controls which objects are selectable in point and area selection. It is not used when you select objects with function keys or menu items. You specify which objects you want to be selectable by clicking the Select mouse button on the **Palette > Set Select Filter** button and completing the dialog box.

Figure 1-27 shows the Schematic Editor Set Select Filter dialog box with the default settings. The three buttons at the top of the dialog box correspond to the palette menus in the Schematic Editor. You can click on those buttons to get the default settings for the palette menus. The **Reset to MGC Defaults** button and the **Add/Route** button both reset the dialog box as shown in the figure. The select filter settings remain in effect for the current Design Architect session, or until you change the settings in the Setup Select Filter dialog box.
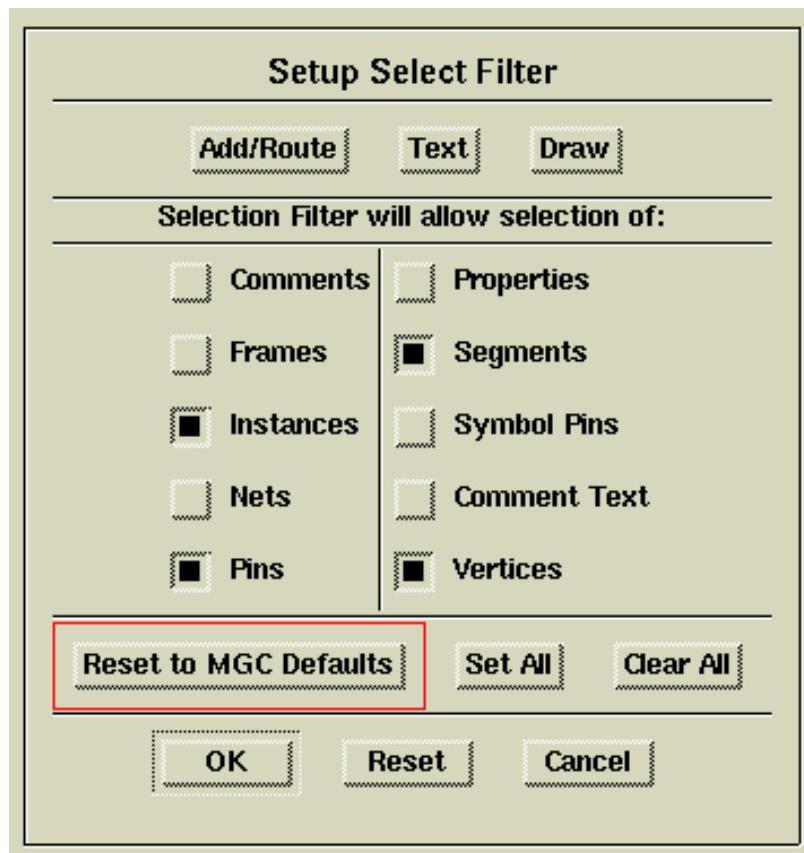
Figure 1-27. Set Select Filter Dialog Box

Using Strokes to View a Sheet

A stroke is a predefined mouse movement, occurring while you hold down the Stroke mouse button (default is center button), that executes one or more functions. Each stroke maps to a pattern on the nine-part stroke grid shown in Figure 1-28.

1	2	3
4	5	6
7	8	9
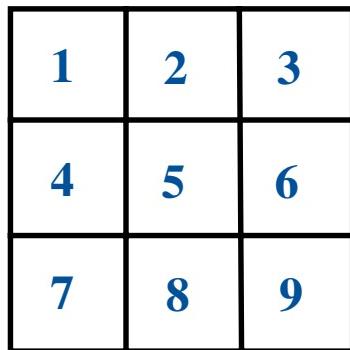
Figure 1-28. Stroke Grid

Most Mentor Graphics applications have predefined mouse strokes. To see the predefined strokes in any Mentor Graphics application, you can either execute the Help stroke, shown in Figure 1-29, or choose the **Help > On Stroke**s pulldown menu item. This displays a message box with diagrams of the strokes available in the type of edit window you are using.

A. Start here.

B. Hold down Stroke mouse
   button and drag mouse.

C. Release mouse button.
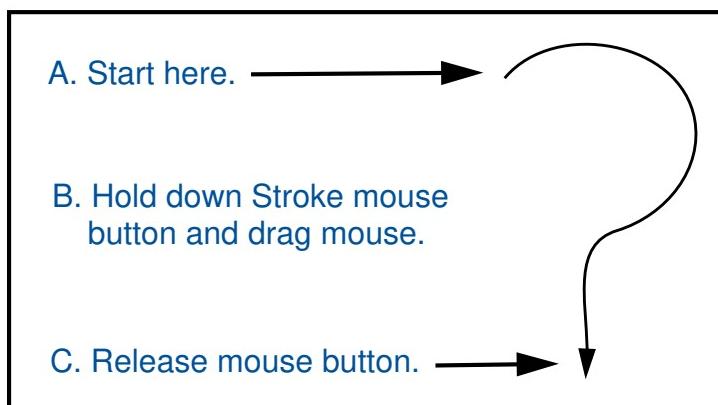
Figure 1-29. Help Stroke

To execute a stroke, hold the Stroke mouse button down as you move the mouse in the pattern that matches the stroke name. For example, the View Area stroke pattern is 1-5-9, and displays the rectangular area defined by the diagonally opposite beginning and ending points of the stroke. The View All and the View Area strokes are shown in Figure 1-30.

C
B
A
View All (9-5-1)

A. Hold down Stroke
   mouse button here.

B. Drag mouse

C. Release Stroke
   mouse button here.

A
B
C
View area (1-5-9)
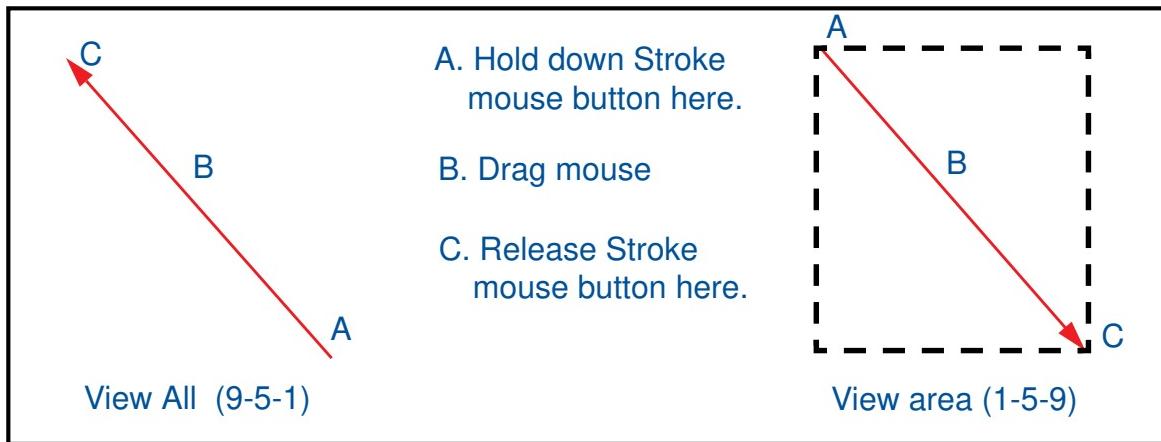
Figure 1-30. View All and View Area Strokes

Lab Exercise

In this lab exercise, you open a schematic sheet and practice viewing areas of the sheet. You also open down into a component. Upon completion of this lab exercise you should be able to:

- Enter the Schematic Editor.

- Select various objects.

- Look at the selection sensitive popup menus.

- Use strokes to view portions of a sheet.

- Open down into a component.

- Create a symbol.

Turn to Module 2—Lab 1: "Introduction to Design Architect with PCB Personality Module".

Lab 1
Introduction to Design Architect with PCB Personality Module

Introduction

There are usually several methods for completing any given task using Mentor Graphics applications. The instructions in this lab generally show only one method, so you can concentrate on the task rather than trying to learn all the ways to accomplish that task. As you become more comfortable with Mentor Graphics applications, you should try other methods to find which one is the most efficient for you.

These lab instructions use mostly menus. If the instructions show the menu for a task, use the menu, even if you see a function key or palette icon that is labeled to do the same function. Sometimes menus, function keys, and palette icons labeled to do the same thing behave slightly differently. If you use a different method for a task, following steps may not work as the lab requires.

In this lab exercise, you open a few sheets in Design Architect, look at different areas on a sheet using strokes and function keys, and look at the hierarchy of the design. Upon completion of this lab exercise, you should be able to:

- Invoke the Schematic Editor.

- View areas of a sheet using strokes and function keys.

- Select design objects.

- Open down into a component.

- Create a symbol.

Procedure

Preparation for Lab

In this part of the lab exercise you invoke Design Architect.

1. If you or your instructor have not already done so, complete the Installation Procedure in the About This Training section of this manual.

2. Invoke Design Manager from a shell, if it is not already displayed.

    *Sys V> **$MGC_HOME/bin/dmgr***

3. Invoke Design Architect by double-clicking the Select mouse button on the Design Architect icon in the Tools window.

    Design Architect opens in a new window.

4. Click the **Maximize** button in the upper right-hand corner of the Design Architect Session window.

    If you are using a Sun workstation, enlarge the window by either dragging a corner of the window to the desired size, or by choosing the **Window > Full Size** menu item.

Activating Windows

Because you can have multiple edit windows open at once in Design Architect (and each window can be for a different editor), it is important to be aware which window is active when you perform an action. Any action you perform, such as saving data, is performed in the active window. Any setups or saves you perform also apply only to the active window.

You can activate a window using any mouse button. However, if you click the Select mouse button (left button by default) in a window, you might accidentally select something while activating the window. If you click the Menu mouse button (right button by default) , you momentarily see the popup menu while activating the window.

Clicking the Stroke/Drag mouse button (center button by default) performs no action (if you do not move the mouse), but it activates the window.

1. Click on each area of the Design Architect Session window.

   Figure 1-31 shows the areas of the Design Architect Session window, and two Schematic windows within the Session.
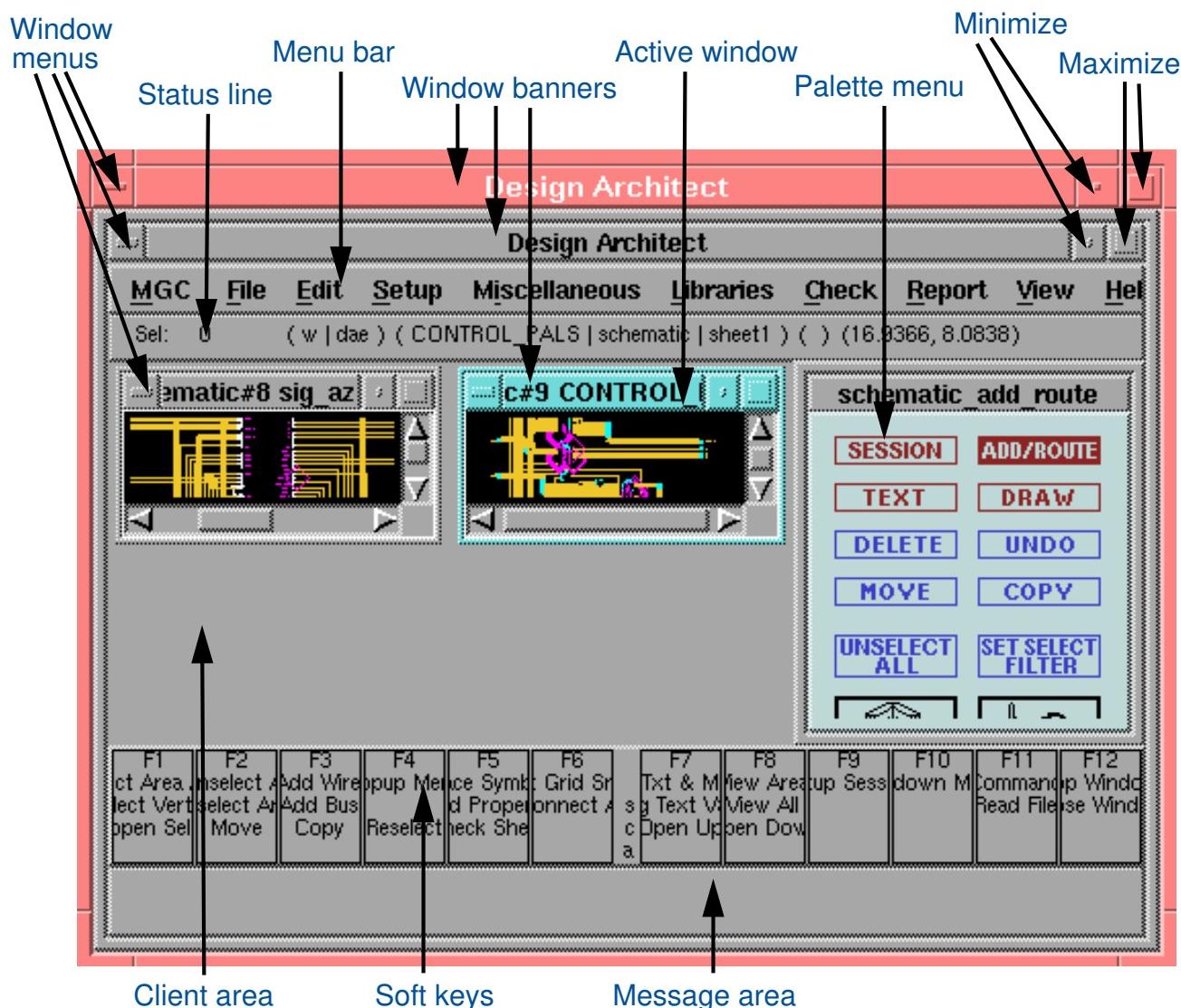
Figure 1-31. Design Architect Session Window Areas

2. Activate each of the windows.

   Use each of the mouse buttons to see what happens in each area. For example, click on the window banner, the soft key area, the message area, and the client area. Notice which of these areas activate the Session by watching the color of the window banner and which palette menu is displayed.

   As you go through the lab exercise, refer to Figure 1-31 if you forget where window items are located.

Opening a Schematic Sheet

1. When Design Architect is ready, click the Select mouse button on the **Palette > OPEN SHEET** icon.

   The Open Sheet dialog box is displayed, prompting you for a component name. The component name to Design Architect is the same as the design name to Board Station applications. Design Architect will recognize the component object as having schematic information, and will ignore any other design objects that are also in that directory. You do not need to type a pathname; you will use the Navigator to find it.

2. In the Open Sheet dialog box, click the Select mouse button on the **Navigator** button. When the Navigator is displayed, use the **Goto** button to navigate to the name of the lab exercise design, **sig_az**.

   The pathname to the component is:

   your_path*/training/board_new/mod2/sig_az*

3. In the Navigator, select the design object, **sig_az** by moving the cursor to the name and clicking the Select mouse button, then **OK** the Navigator.

   The pathname to the **sig_az** design object is automatically placed in the Component field of the Open Sheet dialog box.

4. **OK** the Open Sheet dialog box to open the schematic.

   Sheet1 of the schematic for the component **sig_az** appears in a Schematic Editor window within the Design Architect Session window, as shown in Figure 1-32.

   A component can have many schematic sheets. By default, sheet1 is opened. If you want to open a different sheet, supply the sheet name in the Open Sheet dialog box. If you want to open several sheets of a component at once, repeat this process for each sheet you want to open, supplying a different sheet name each time.
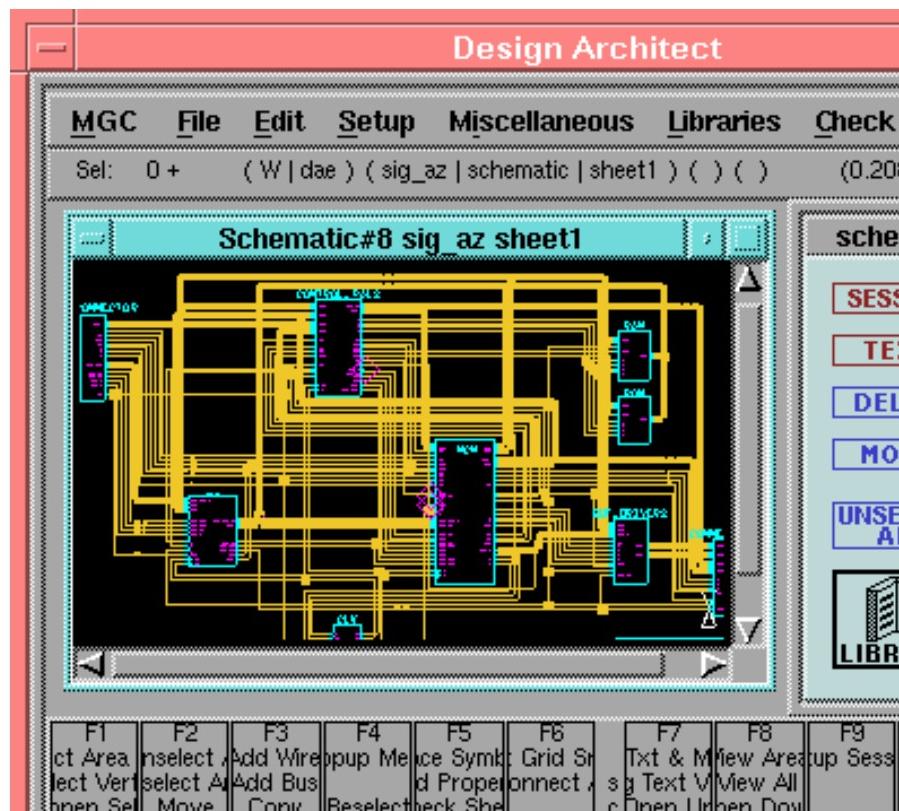
Figure 1-32. Sheet1 Displayed in a Schematic Window

Click the **Maximize** button on the Schematic window so it fills the client area inside Design Architect.

If you are using a Sun workstation, enlarge the window by either dragging a corner of the window to the desired size, or by choosing the **Window > Full Size** menu item.

5. With the cursor inside the Schematic window, click the Stroke mouse button to activate the window (window banner highlights).

Viewing

The viewing (View All and View Area) strokes are the same in Design Architect as they are in Board Station tools. In this section you will practice using the View All and View Area strokes. The default, the Stroke key is the center mouse button.

1. View all of the design using the View All stroke shown in Figure 1-33.
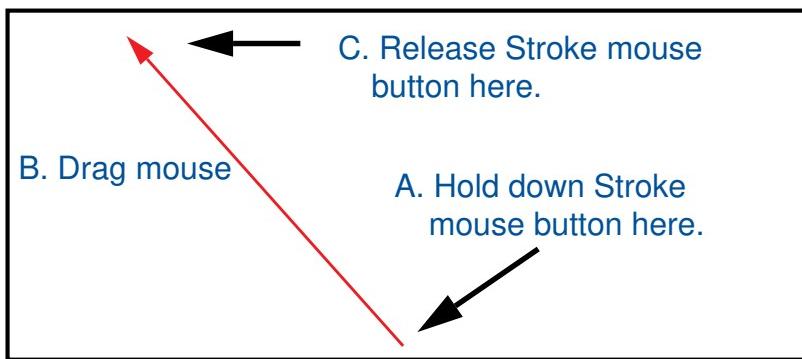
Figure 1-33. View All Stroke (9-5-1)

2. Use the View Area and View All strokes to look at different areas of the schematic sheet. The beginning and end points of the View Area stroke define the viewing area, as shown in Figure 1-34.
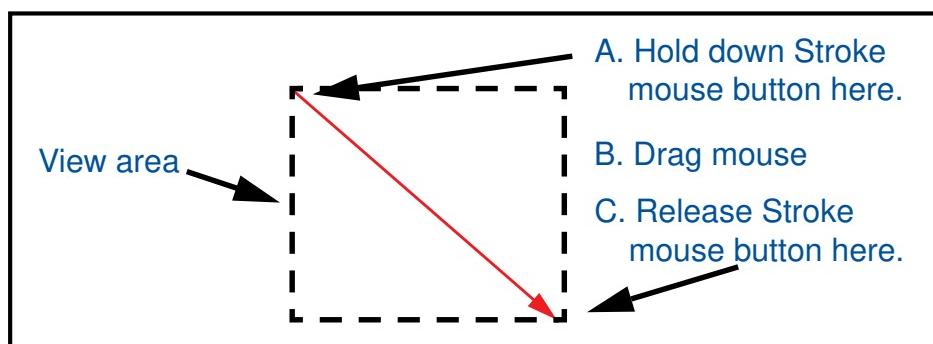
Figure 1-34. View Area Stroke (1-5-9)

Regarding viewing an area or viewing all of a design, you should notice that the icons in Design Architect windows are not identical to those you find in Board Station tools. In Design Architect (as well as in Board Station tools) you will probably find that the View All and View Area functions are most conveniently performed by using strokes or the View All and View Area function keys.

Selecting Design Objects

Now you will practice selecting design objects on the sheet, first, by clicking on them, next, by defining a selection area, and finally, by choosing a menu item.

1. Move the cursor to one of the symbol instances on the open schematic and click the Select mouse button.

    The instance should be highlighted to show it is selected. The first item in the status line directly below the pulldown menu bar shows how many objects are selected. In Figure 1-35, *Sel: 1* indicates one object is selected. + means the selection set is open; when you click on objects to select them, they are added to the selection set until an edit operation is performed.
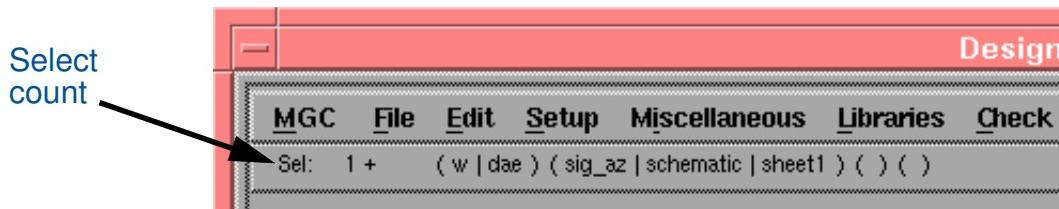
Figure 1-35. Select Count in Status Line

2. Move the cursor to another symbol instance and click the Select mouse button.

    Two instances should be highlighted, and the status line should show *Sel: 2+* to indicate two objects are selected and the selection set is still open.

3. Unselect all objects by pressing the Unselect All function key.

4. Select different design objects such as pins and nets by clicking the Select mouse button on them.

5. Click the Select mouse button on a highlighted (selected) object.

   The object is no longer selected, and the select count shows the new number of selected objects. If you click on the same object again, it is reselected.

   As you select and unselect objects, look at the popup menus by pressing the Menu mouse button. The popup menus are *selection sensitive* which means the selected objects determine which popup menu is displayed. Table 1-2 shows which menu is displayed when particular objects are selected.

   Some functions, such as Move and Copy, are accessible through all the popup menus. The other items in each menu are those you most likely need for the design objects you have selected.

Table 1-2. Selection Sensitive Popup Menus

Popup Menu	Selected Objects
Net	Nets and/or instance pins
Instance	Instances
Property/Text	Property and/or comment text
Draw	Comment graphics and/or symbol pins
Mixed Selection	Combination of types of objects
Add	Nothing selected

6. Unselect all objects by pressing the Unselect All function key.

7. Move the cursor just outside of the upper left-hand corner of a symbol instance. Press and hold the Select mouse button as you move the cursor just outside of the lower right-hand corner of the instance, then release the mouse button.

Figure 1-36 shows the upper left and lower right corners of a selection rectangle, and the resulting selection highlighted. The select count includes the symbol instance, pins, net segments, and vertices.
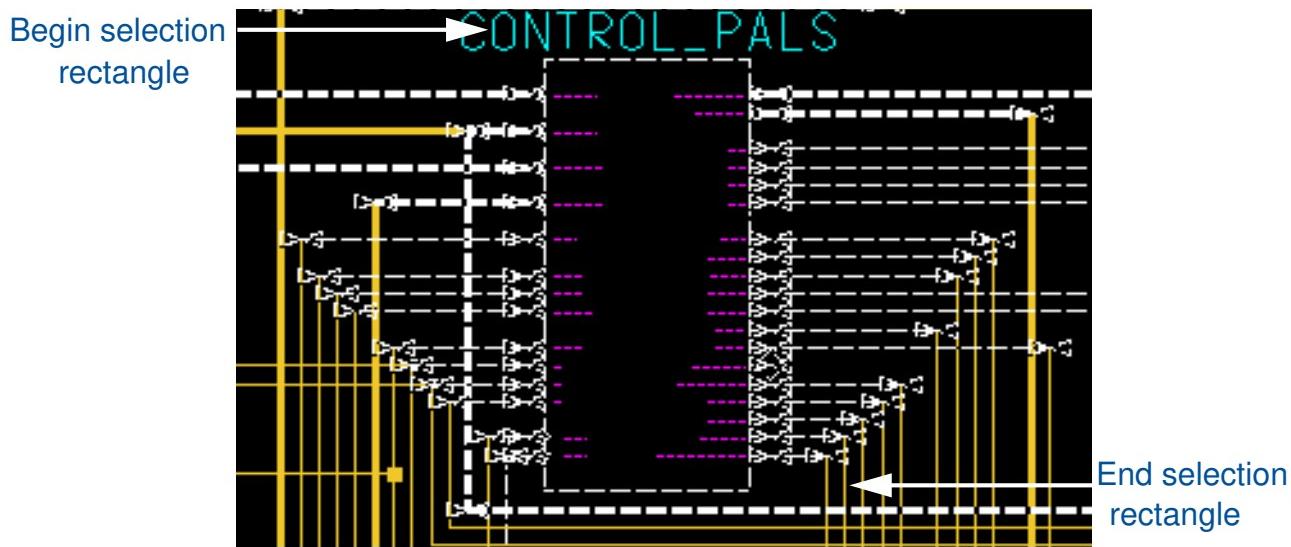
Figure 1-36. Selecting an Area

Using the Select Filter

The select filter determines which objects can be selected using the mouse. You can change the select filter when you want to select only certain objects.

1. Click the Select mouse button on the **Palette > Set Select Filter** button.

   The Set Select Filter dialog box, shown in Figure 1-37, is displayed for you to specify the type(s) of objects that you want to select.
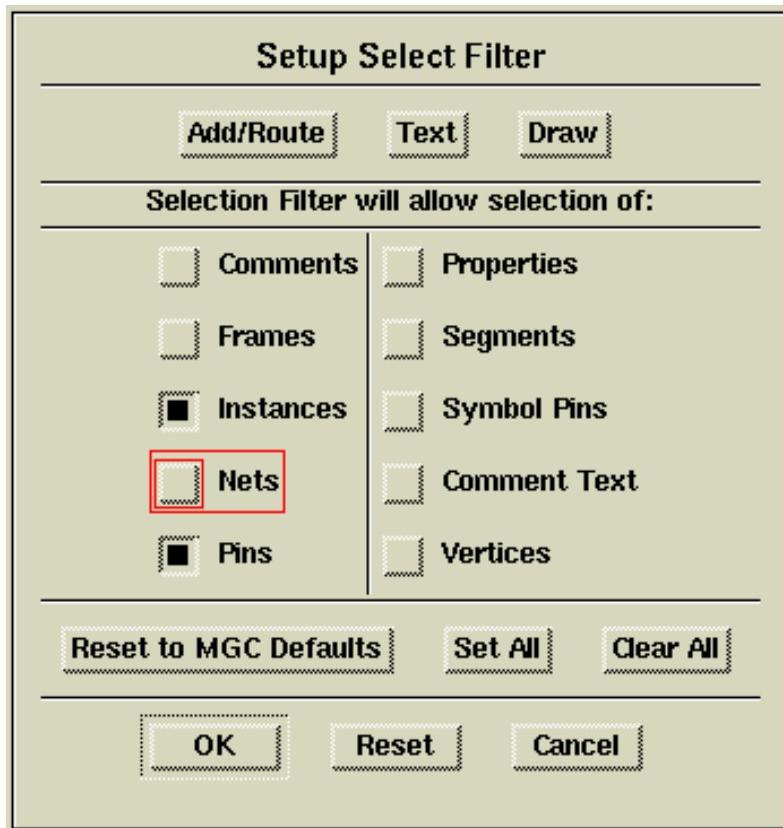
Figure 1-37. Set Select Filter Dialog Box

2. In the dialog box, click the buttons as needed to highlight only the **Instances** and **Pins** buttons, then **OK** the dialog box.

3. Unselect all objects by pressing the Unselect All function key.

4. Define a selection rectangle using the Select mouse button, as you did previously.

   This time, only instances and pins within the rectangle are selected.

5. Click on the **Set Select Filter** button to display the dialog box again.

6. Click on the **Add/Route** button and **OK** the dialog box.

   The select filter is now reset to the default for most schematic tasks. You could have clicked on the **Reset to MGC Defaults** to do the same thing. The **Text** button resets the filter to select properties and comment text; **Draw** resets the filter to select comment objects and symbol pins. The select filter does not control the objects selected through menu items, except the **Select Area > Anything** menu item.

7. Unselect objects by pressing the Unselect All function key.

Making the Selections

1. Choose the **[Add] Select Area > Pin** menu item.

   The brackets around the menu name (**[Add]**) indicate the name of the popup menu. The name changes because these menus are selection sensitive.

2. When the Select Area prompt bar appears, use the mouse to define a selection area around a symbol instance.

   Only the pins on the symbol instance are selected.

3. Unselect all objects, then choose the **[Add] Select > All > Pin**.

   All pins on all symbol instances are selected. When you press the Menu mouse button, notice that the **Net** popup menu now appears.

4. Execute the View All stroke.

5. Choose the **[Net] Select > All** menu item.

   Everything on the schematic is selected.

6. Choose the **[Mixed Selection] Unselect > Area > Net** menu item.

7. When the Unselect Area prompt bar appears, press and hold the Select mouse button as you move the cursor to define a large rectangle approximately in the center of the schematic.

Figure 1-38 shows diagonally opposite corners of the rectangle, and the remaining selected objects. Notice that nets completely within, and partially within, the rectangle are unselected.

Begin rectangle
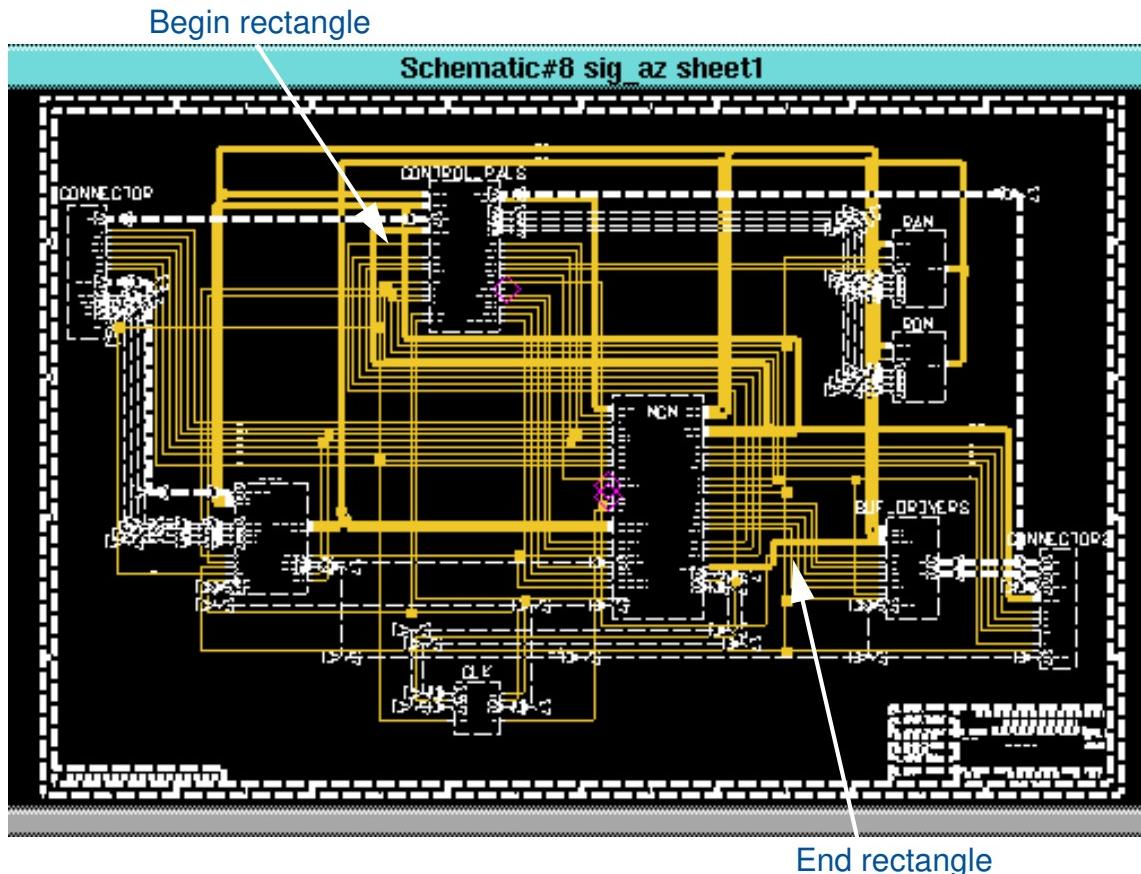
End rectangle

Figure 1-38. Unselecting Nets in an Area

8. Unselect all objects, then choose the **[Add] Select > Exterior > Anything**.

9. When the prompt bar appears, use the mouse to define a rectangle from the upper left corner of the sheet border to the lower left corner of the title block.

Notice that any object partially outside the rectangle was selected.

10. Choose the **[Mixed Selection] Unselect > All** popup menu item.

11. Click the **Minimize** button of the Schematic window.

This reduces the window to an icon without closing it. To iconify a window on a Sun workstation, click on the **Window** menu button.

Opening Down into a Component

You will restore the design sheet icon to a window, view an area of the schematic containing the CONTROL_PALS component, and open down into the component.
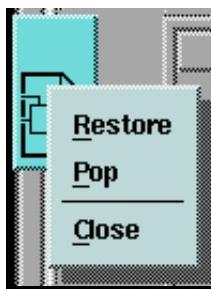
1. Click the Menu mouse button on the design sheet icon in the Session window, then choose **Restore** from the popup menu that appears by the icon.

2. View the area surrounding the CONTROL_PALS component on the left side of the schematic sheet using the View Area stroke. Refer to Figure 1-34 on page 1-40, if necessary.

3. Click the Select mouse button on the CONTROL_PALS component. If the select count is more than one object, press the Unselect All function key, then select the component again.

4. Choose the **File > Open Down > Choose Model** menu item.

This displays the Open Down dialog box, illustrated in Figure 1-39.
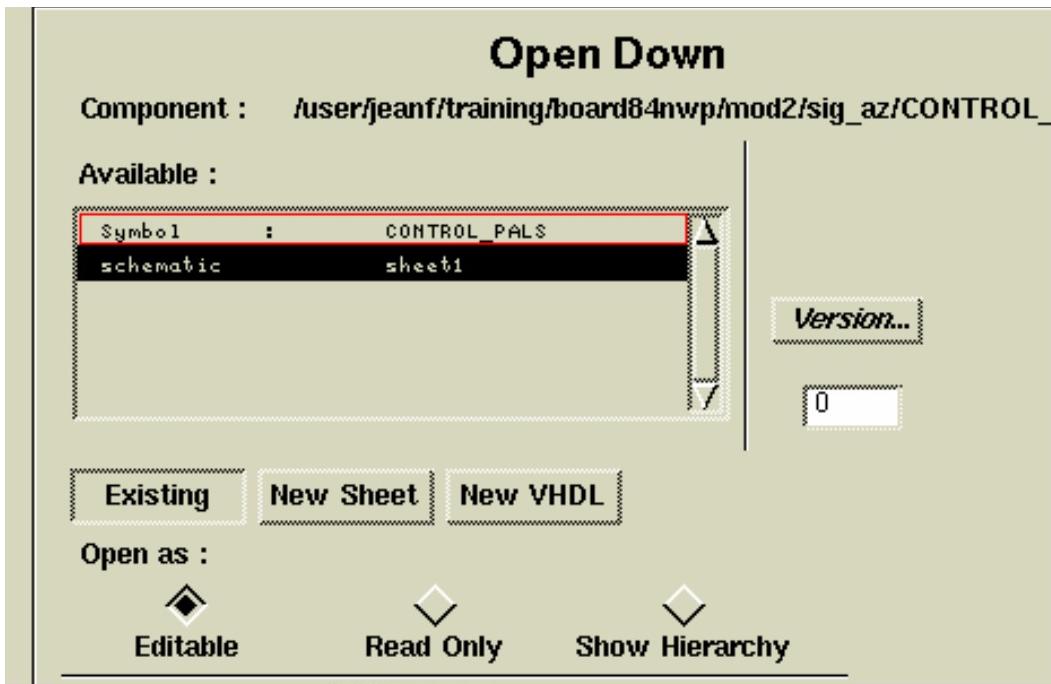
Figure 1-39. Open Down Dialog Box

5. Choose *schematic* from the list of available models, and **OK** the dialog box.

   Version 0 indicates the most current version of the sheet will be opened. Design Architect opens another Schematic window to display the sheet for the CONTROL_PALS component.

   You can view the hierarchy of a design using this method of selecting a component and opening down. Whenever you choose *schematic* in the dialog box, the schematic represented by the selected component is displayed in a Schematic window. If you choose *symbol* from the list of models, that symbol is opened for editing in a Symbol window. The component is primitive if there is no schematic listed in the dialog box.

6. Choose the **MGC > Setup > Session** pulldown menu item.

   The Setup Session dialog box is displayed for you to choose the window areas to display, the mouse button double-click speed, and window arrangement within the client area of Design Architect.

7. Click the Select mouse button on the **Quadrant Tiling** button under Window Layout.

   The open Schematic windows are redrawn so each one occupies one quarter of the client area.

Opening Multiple Views of a Sheet

Instead of opening the **sig_az** sheet and opening down into the CONTROL_PALS component as you did earlier, you could have opened that schematic sheet directly using the Navigator in the Open Sheet dialog box. The pathname for this component is:

your_path*/training/board_new/mod2/sig_az/CONTROL_PALS*

1. Activate the Session window by clicking the Stroke/Drag mouse button in the client area.

2. Click on the **Palette > OPEN SHEET** icon.

3. Choose the CONTROL_PALS component in the Navigator, and **OK** the Open Sheet dialog box.

   Another Schematic window is opened on the CONTROL_PALS component.

4. On the **sig_az** sheet, select an area surrounding the CONTROL_PALS component.

   Does this action select any objects in the other two windows?
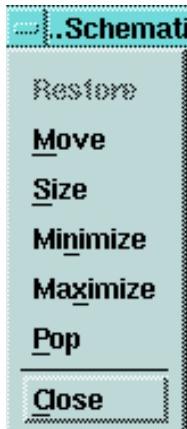
5. Select various objects in one of the CONTROL_PALS component windows.

   Are any objects in the other windows selected by this action?

6. Close all schematic windows by choosing the **Window > Close** menu item in each window. Click **NO** on the dialog box that asks if changes to the sheet should be saved. Do not close the Design Architect session.

Creating a Symbol

In this part of the lab exercise, you create a symbol.

1. Open a new symbol sheet by clicking on the **Palette > Open Symbol** icon.

   This displays the Open Symbol dialog box shown in Figure 1-40.
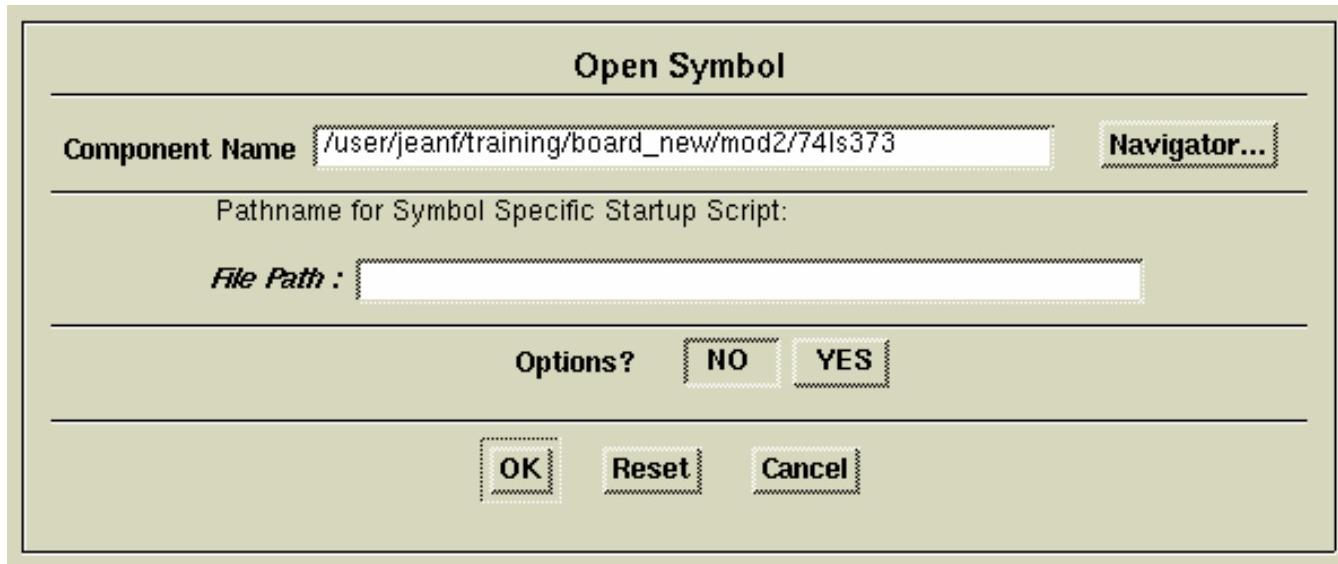
Figure 1-40. Open Symbol Dialog Box

2. Enter the following pathname in the **Component Name** text entry box:

   your_path**/training/board_new/mod2/74ls373**

   and click the **OK** button on the dialog box.

   A Symbol Editor window opens with the default grid. The light dots are the snap grid, and the heavy dots are the pin grid.

Create the Symbol Body

Refer to Figure 1-41 while drawing the symbol body.
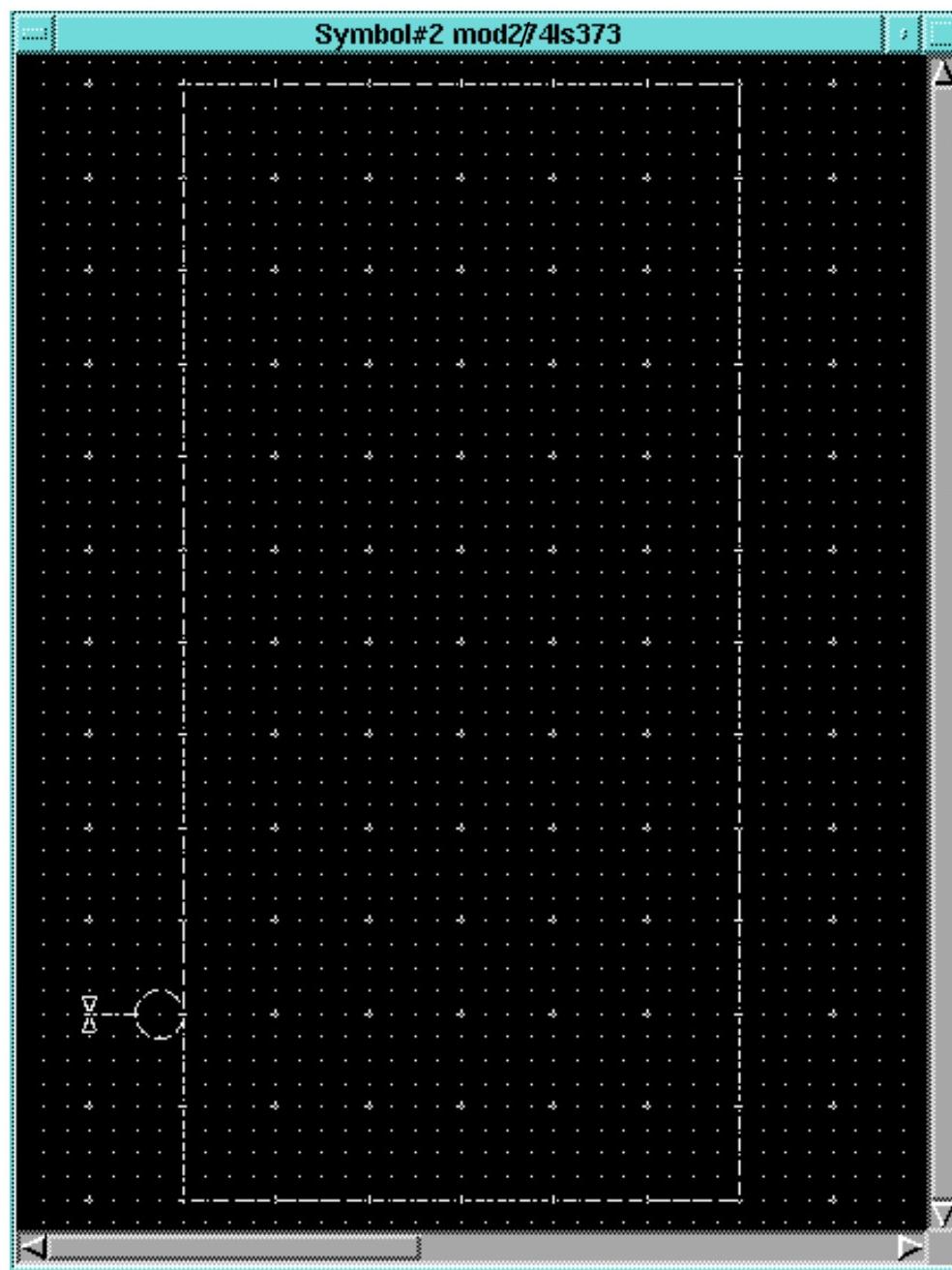
Figure 1-41. Drawing the Symbol Body

1. Click on the **Symbol_Draw Palette > Add Rectangle** icon.

   The Add Rectangle prompt bar appears, prompting you for a location.

2. With the cursor on one of the pin grid points in the Edit window, press the Select mouse button to define one corner of a rectangle. Hold the mouse button down as you move the cursor six pin grid points to the right, and 12 pin grid points down from the starting point, release the mouse button.

3. Click on the **Symbol_Draw Palette > Add Circle** icon.
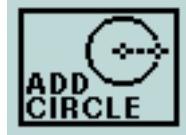
   A prompt bar appears prompting you for the center point and the radius of the circle.

4. Position the cursor two pin grid points up from the bottom edge of the rectangle and one snap grid point to the left. Press the Select mouse button, move the cursor one snap grid point to the right (touching the rectangle), and release the mouse button.

   Next you draw the whisker for this pin.

5. Choose the **[Symbol Body & Pins] Draw > Two Point Line** popup menu item.

6. Place the cursor on the pin grid point to the left of the circle. Press the Select mouse button, move the cursor so it touches the circle, then release the mouse button.

Add Symbol Pins

Refer to Figure 1-42 for the steps in this subsection.
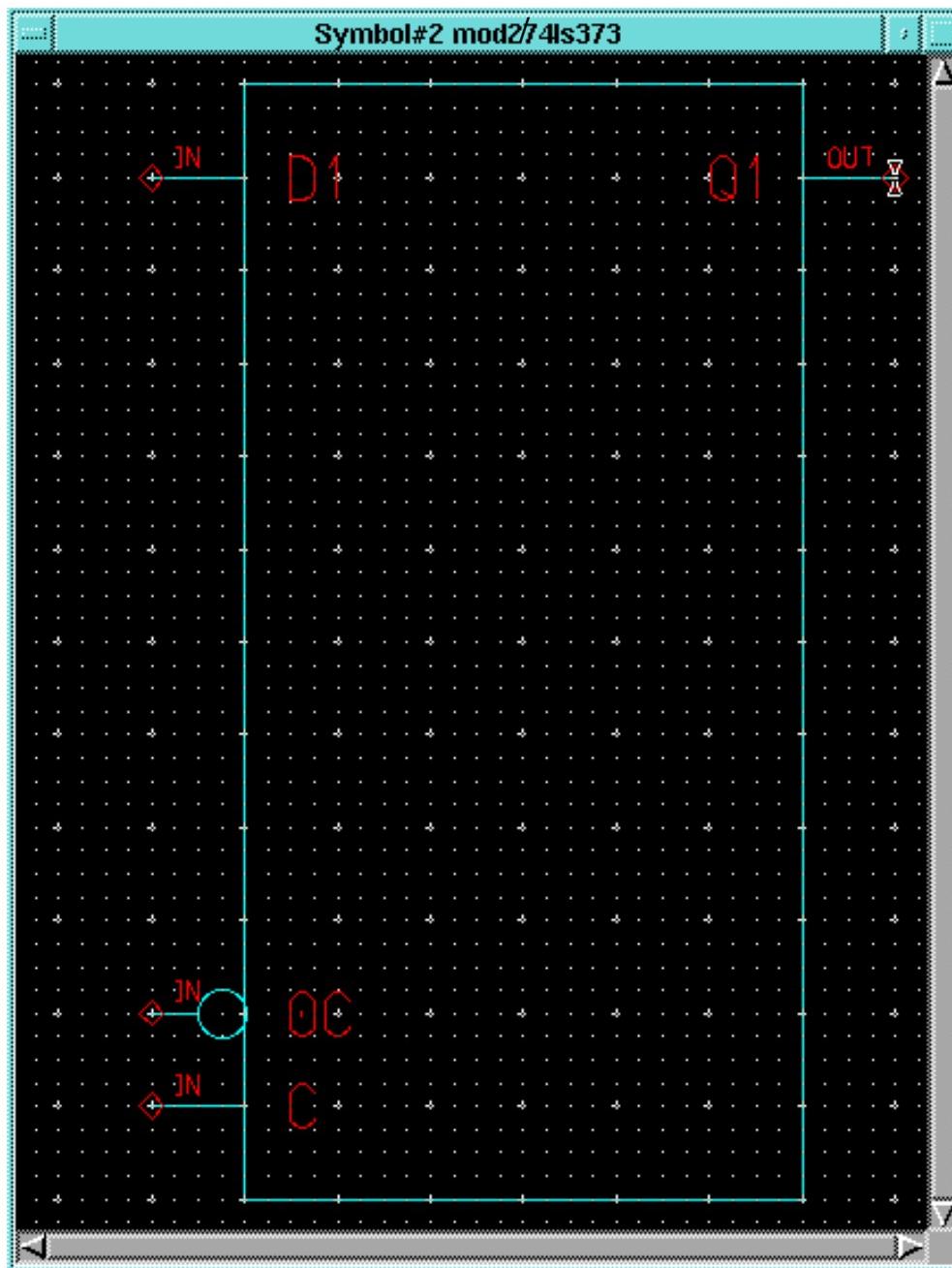
Figure 1-42. Adding Symbol Pins

1. Click on the **Symbol_Draw Palette > Add Pin** icon.

2. Complete the Add Pin dialog box as shown in Figure 1-43 and **OK** the dialog box. When the prompt bar appears, click the Select mouse button on the end of the whisker you just drew.

A pin is placed on the end of the whisker. When the symbol is placed on a schematic and a wire is connected to the pin, the diamond will disappear, indicating that there is a connection. The **0C** is the Pin property text, and the **OUT** is the Pintype property text. Do not be concerned about the text placement; you will move it later.
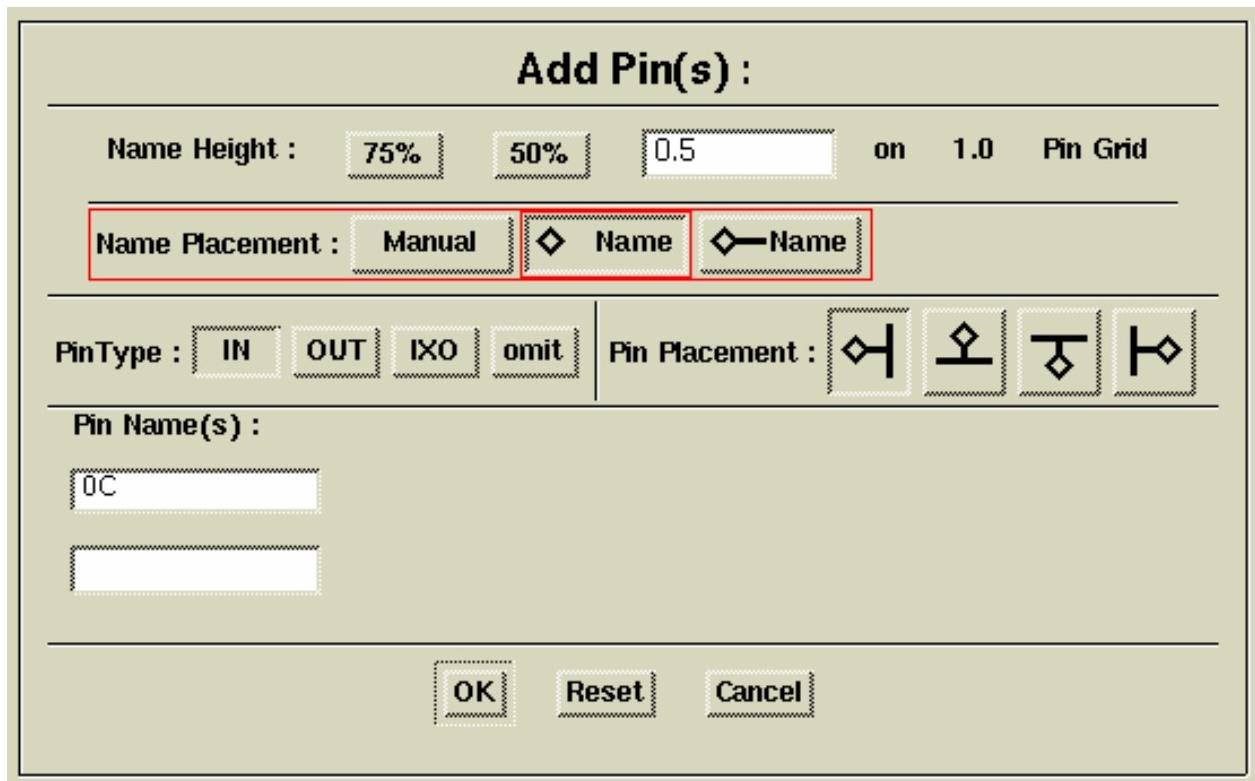
Figure 1-43. Adding the 0C Pin

3. Click on the **Symbol_Draw Palette > Add Pin** icon again, and complete the Add Pin dialog box as shown in Figure 1-44.
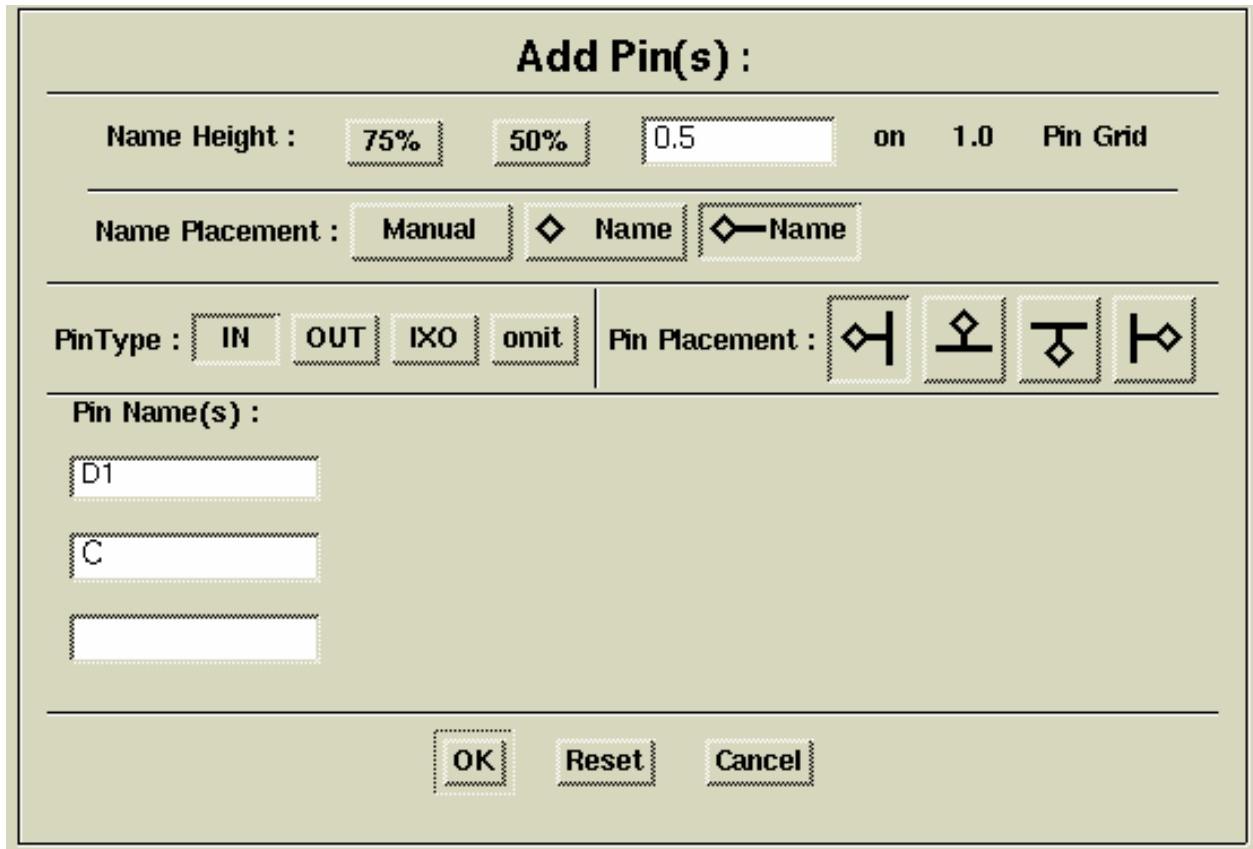
Figure 1-44. Adding the C and D1 Pins

4. When you are prompted for pin locations, click the Select mouse button on the pin grid points for pins D1 and C, respectively, as shown in Figure 1-42 on page 1-53.

   You need to leave space for the whisker, and the pins must be on pin grid points. Now you add one pin on the right side of the symbol.

5. Click on the **Symbol_Draw Palette > Add Pin** icon, and complete the Add Pin dialog box as shown in Figure 1-45.
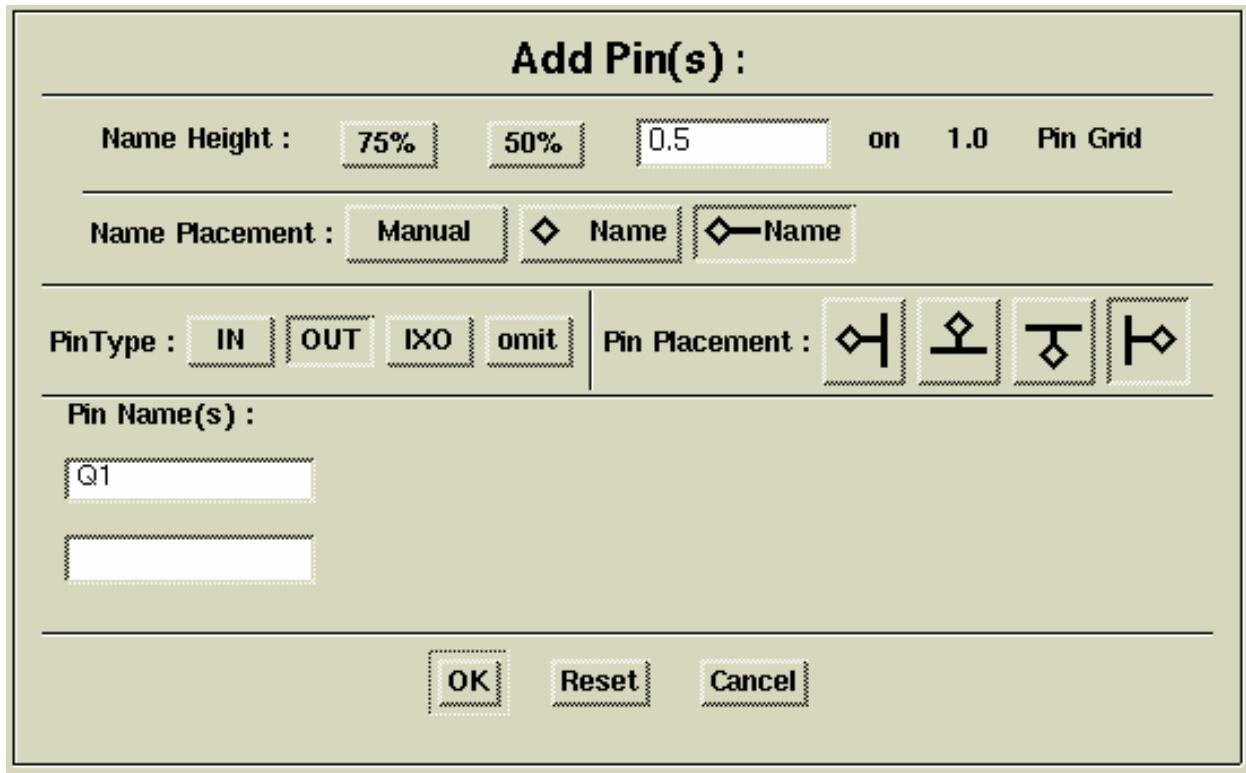
Figure 1-45. Adding the Q1 Pin

6. Press the Unselect All function key.

7. Choose the **[Add] Select > Area > Property** popup menu item, then click the Select mouse button on the **0C** text.

8. Click on the **Symbol_Draw Palette > Move** icon. Move the selected text so that it lines up with the **C** and **D1** pin names as shown in Figure 1-42 on page 1-53.

9. Press the Unselect All function key.

You could have used the same method to add all the symbol pins. Instead, you learn another method in the next subsection.

Copy Pins

The method you use to add the rest of the pins is convenient when you have many similarly named pins on the same side of the symbol body. You set the select filter first, then select and copy the pins on each side of the symbol.

1. Click on the **Palette > Set Select Filter** icon. When the dialog box appears click on the **Reset to MGC Defaults** button, as shown in Figure then **OK** the dialog box.
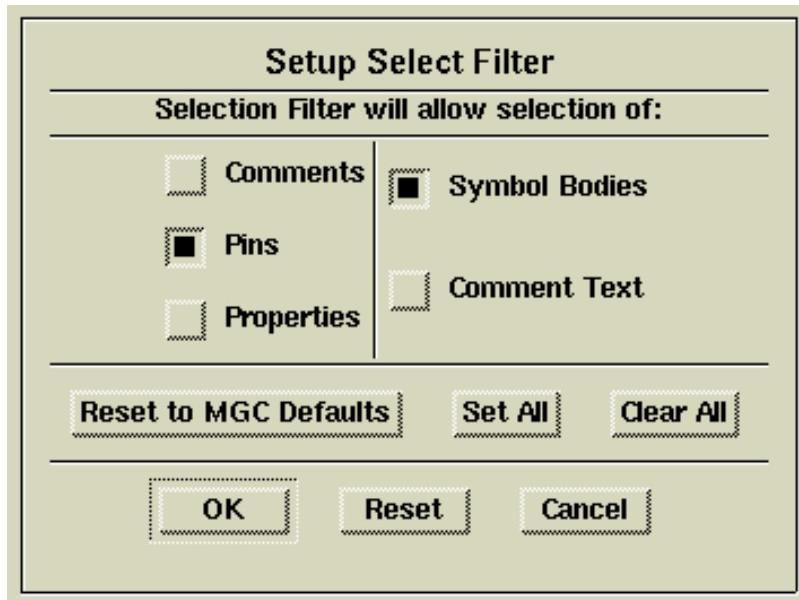
Figure 1-46. Set Select Filter Dialog Box

2. Select the **D1** pin and whisker by pressing the Select mouse button as you move the cursor to form a rectangle around the pin and the end of the whisker.

    Be sure you do not select the symbol body; there should be two objects selected.

3. Choose the **[Symbol Body & Pins] Copy > Multiple** popup menu item.

4. Enter **7** in the prompt bar, as shown in Figure 1-47. **OK** the prompt bar and click the Select mouse button one pin grid point below the **D1** pin.

   Figure 1-47. Copy Multiple Prompt Bar

   Seven copies of the pin, whisker, and property text are placed on the left side of the symbol. You will renumber the pins after creating the rest of the pins on the right side of the symbol.

5. Press the Unselect All function key, then select the **Q1** pin and whisker.

6. Choose the **[Symbol Body & Pins] Copy > Multiple** popup menu item again, enter **7** in the Copy Multiple prompt bar, and **OK** the prompt bar.

7. Click the Select mouse button pin one pin grid below the **Q1** pin.

   Seven copies of the **Q1** pin, whisker, and property text are placed on the right side of the symbol body. Figure 1-48 shows how the symbol looks after copying the pins.
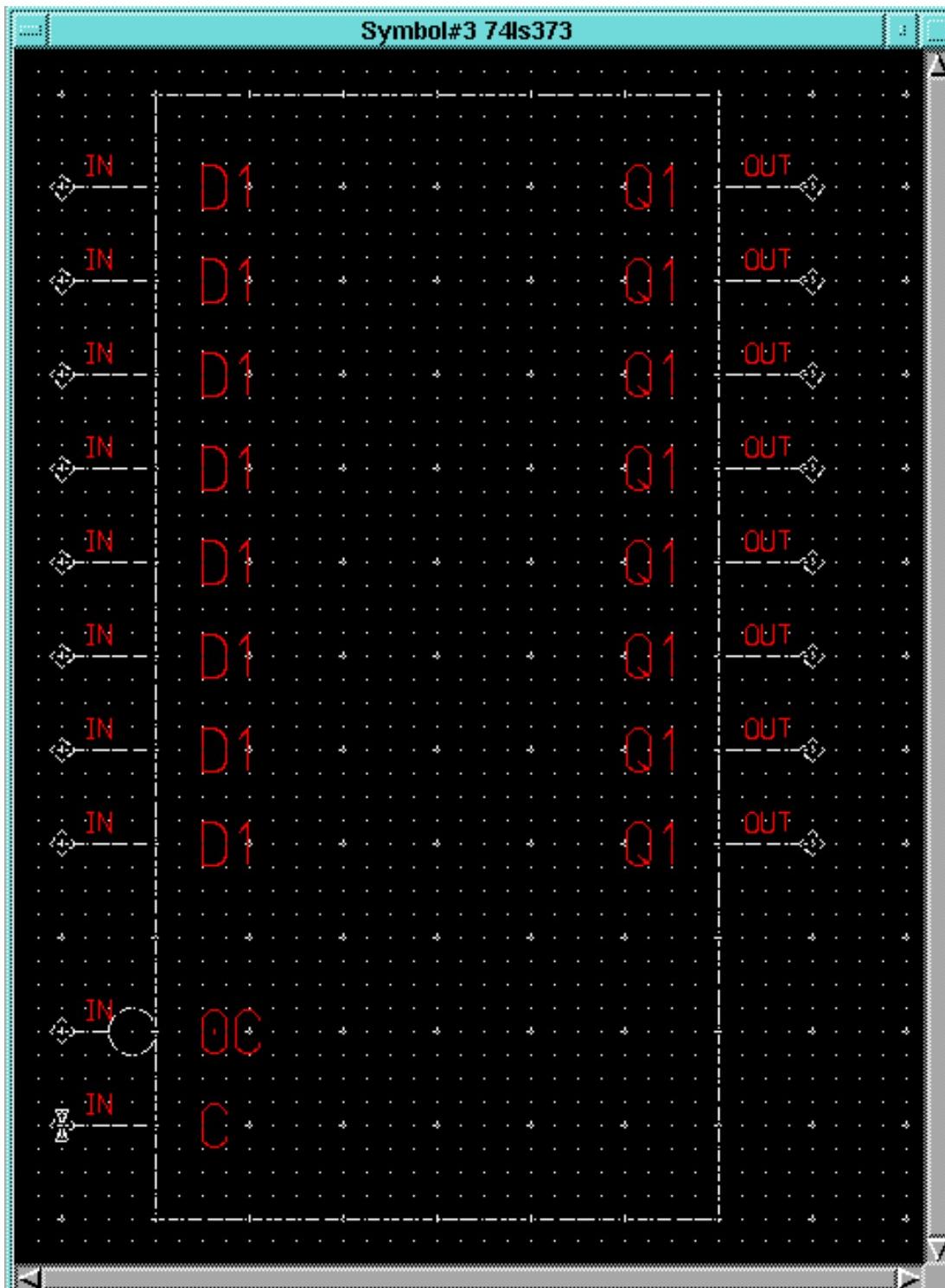
Figure 1-48. Symbol After Copying Pins

Sequence Text

TEXT
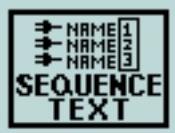
1. Click the Select mouse button on the **Palette > Text** button to change palette menus.

2. Press the Unselect All function key. Choose the **[Property/Text] Select > Area > Property** popup menu item. Hold the Select mouse button down as you form a selection rectangle around the column of **D1** text.

3. Click on the **Text Palette > Sequence Text** icon. Complete the Sequence Text dialog box as shown in Figure 1-49.

   You need to specify **New Prefix: D** so it is not dropped when the text is changed.
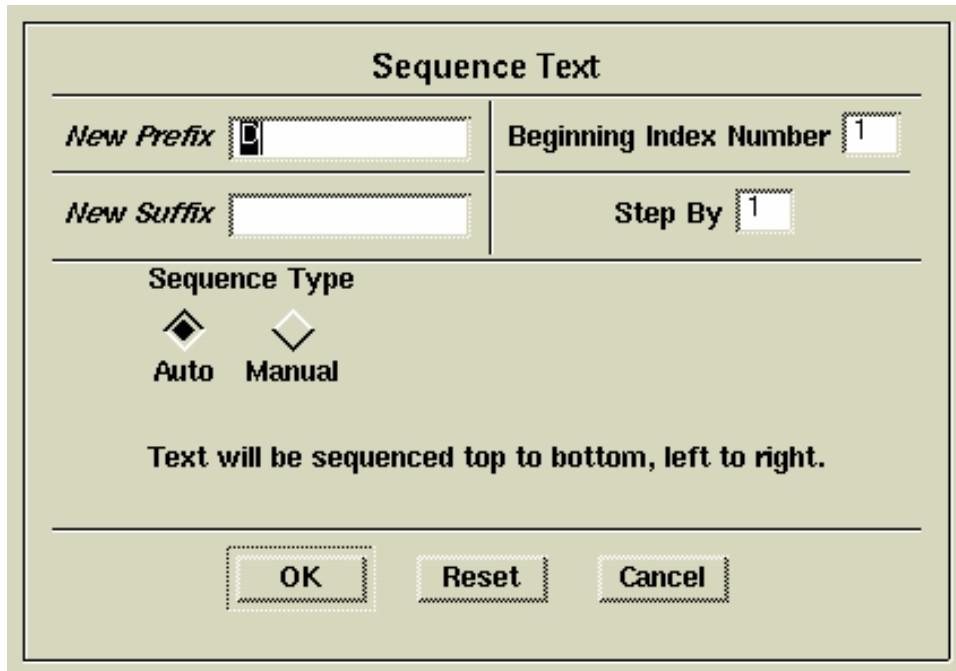
Figure 1-49. Sequence Text Dialog Box

4. Press the Unselect All key. Use the same method to sequence the **Q1** column of text:

New Prefix: **Q**
Beginning Index Number**:   1**

Add Pin Numbers

You need to add pin numbers to the symbol. Properties have not been discussed yet, so some of this section may not be very clear until after the next lesson. A pin number associates a logical symbol pin with a physical pin on a component.

You are going to add the same Pin_no property to all pins, and then renumber them. This is easier and faster than adding the Pin_no property to each individual pin. If there were only a few pins, it would probably be faster to add the property to each pin.

1. Press the Unselect All function key.

2. Choose the **[Add] Select > Area > Pin** popup menu item, and select the pins on the left side of the symbol.

3. Choose the **[Symbol Body & Pins] Properties > Add > Add Single Property** popup menu item.

4. Complete the Add Property dialog box as shown in Figure 1-50. Click the Select mouse button on **PIN_NO** in the list box. When you select a property name from the list box, you do not need to enter a name in the text entry field.

Add Property

Highlighted property name will

be used unless new property

name is filled in below

Existing Property Name

INIT
PIN_NO
RULE
FALL
DRIVE
RISE
PINTYPE
PIN

New Property Name

Property Value 1

Graphic

◆ Graphic

◇ Nongraphic

Property Type

◆ String

◇ Number

◇ Expression

◇ Triplet

◇ Default For This Property Name

Stability Switch

◆ Variable

◇ Fixed

◇ Protected

◇ NonRemovable

Visibility Switch

◇ Visible

◆ Hidden

OK   Reset   Cancel

Figure 1-50. Add Property Dialog Box

5. When you **OK** the dialog box and move the cursor in the edit window, the **1** attached to each selected pin moves with the cursor. Move the text next to the pins and click the Select mouse button.

6. Press the Unselect All function key.

7. Use the same method to add the Pin_no property to the pins on the right side of the symbol.

   Figure 1-51 shows the symbol with Pin_no properties. Next, you will renumber the pins.
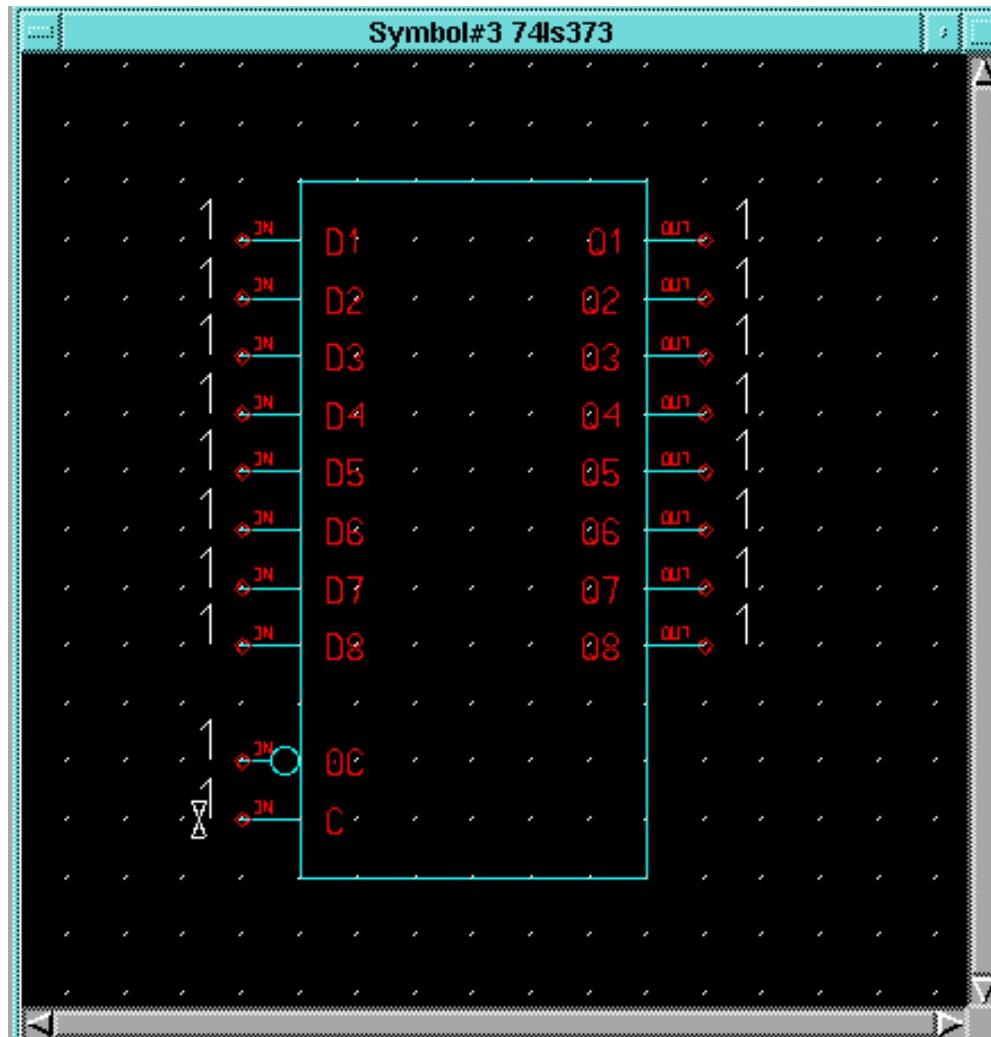
Figure 1-51. Pin Numbers Added to the Symbol

8. Press the Unselect All function key.

9. Choose the **[Add] Select > By Property > Name-Value-Type** popup menu item, and complete the dialog box, as shown in Figure 1-52.
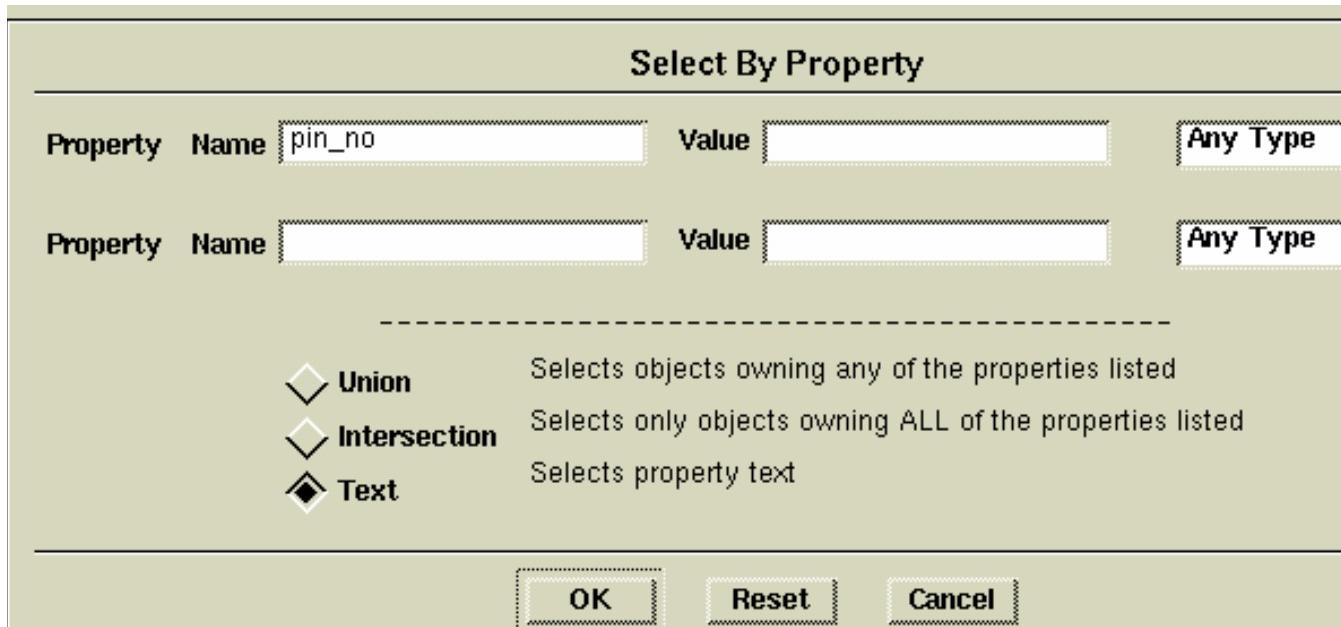
Figure 1-52. Select By Property Dialog Box
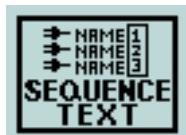
10. Click on the **Text Palette > Sequence Text** icon. Complete the Sequence Text dialog box as shown in Figure 1-53. Be sure to click on **Manual** for **Sequence Type**.
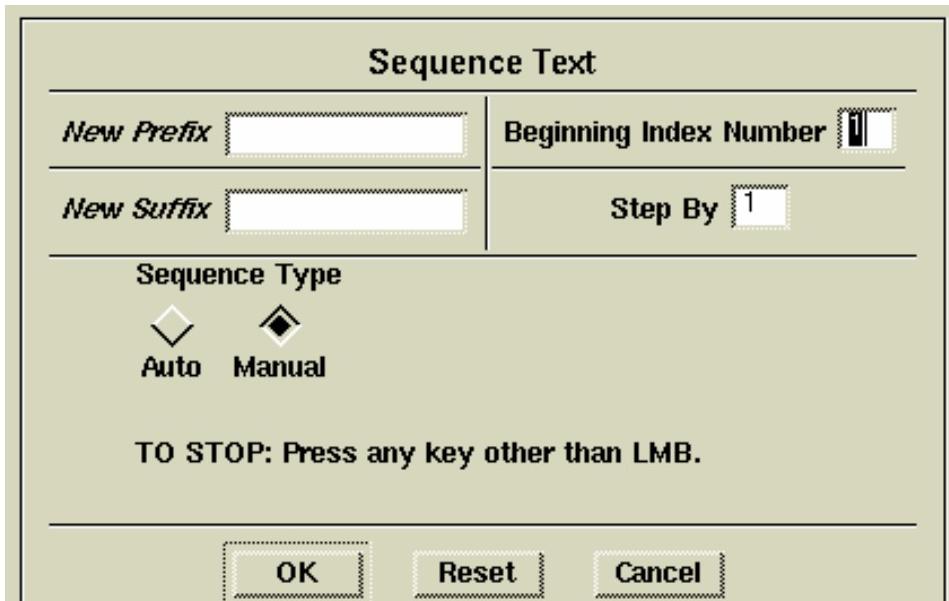
Figure 1-53. Sequence Text Dialog Box for Pin_no Properties

11. Renumber the pins using Figure 1-54 as a guide. Click the Select mouse button on each successive number. The message window tells you what the next number is. There is no pin **10** on the symbol, so change the number for pin **C** from **1** to **10** to **11**. When you have renumbered all the pins, click the right mouse button to stop.
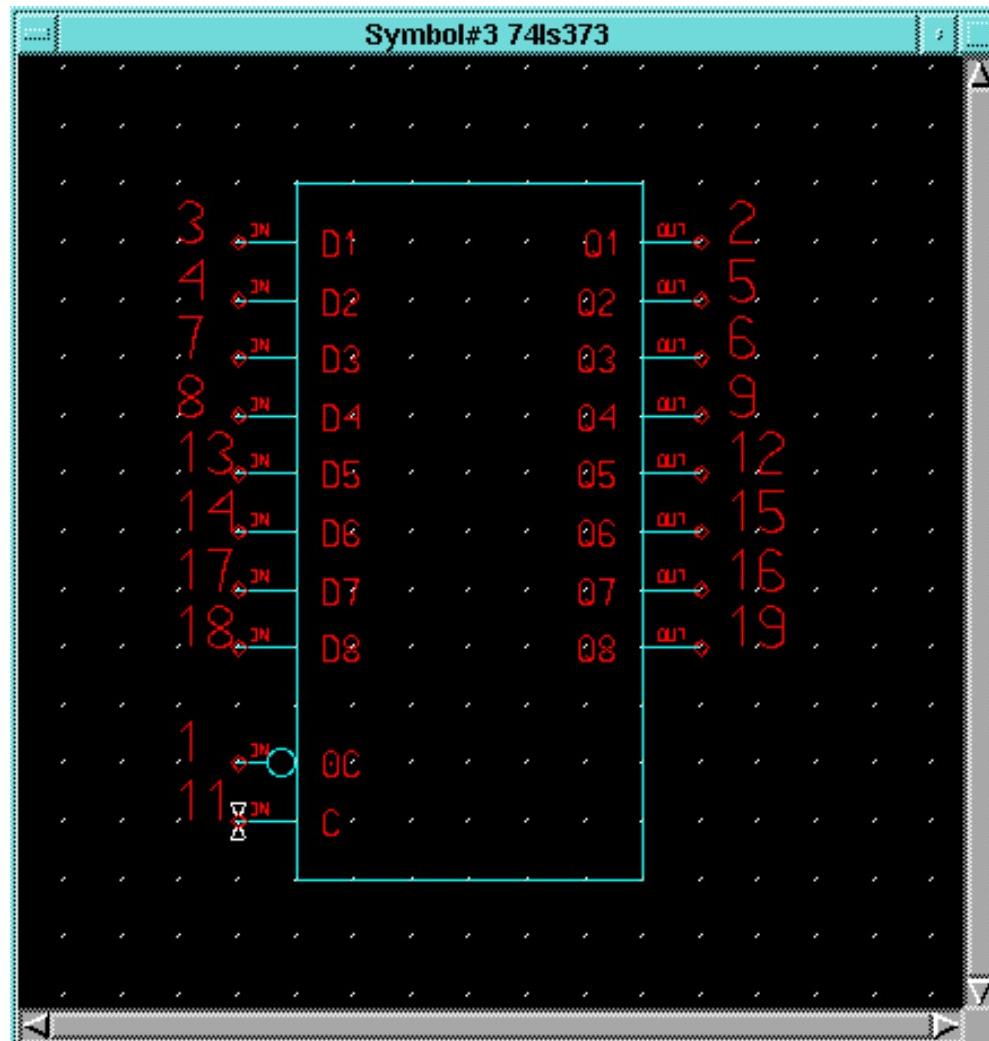
Figure 1-54. Completed 74ls373 Symbol

Normally, you would add other properties to the symbol. After you learn more about properties, you might want to return to this symbol and add more.

Check and Save the Symbol

A symbol must pass some basic checks before you can place it on a schematic sheet.

Choose the **Check > With Defaults** pulldown menu item.

Design Architect checks the symbol and displays the results in a Check Report window, shown in Figure 1-55.


```
Check#1  74ls373:Symbol

Check Symbol "74ls373/74ls373"
Check Body -------   0 errors   0 warnings  (MGC-required)
Check Interface --   0 errors   0 warnings
Check Pin --------   0 errors   0 warnings  (MGC-required)
Check Special ----   0 errors   0 warnings  (MGC-required)

 "74ls373/74ls373" passed check :   0 Errors, 0 Warnings
```
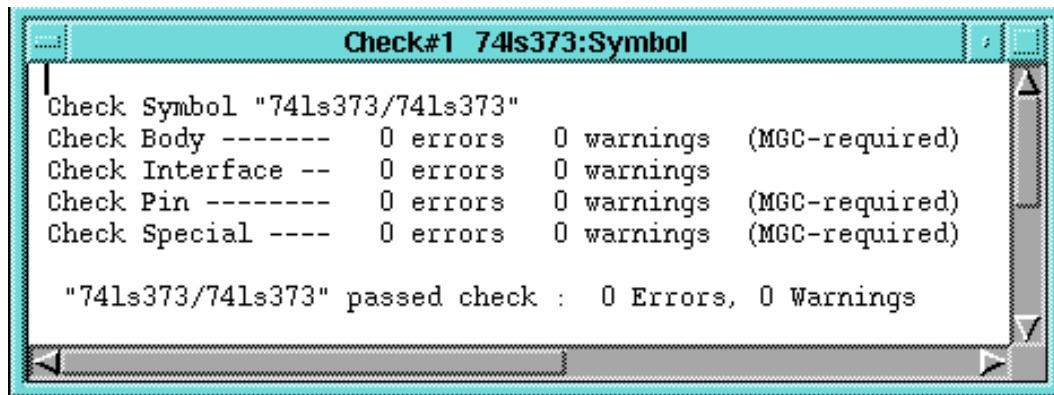
Figure 1-55. Symbol Editor Check Report Window

1. Close the Check Report window by choosing the **Window > Close** menu item.

2. Choose the **File > Save Symbol > Default Registration** pulldown menu.

3. Close the Symbol Editor and Design Architect by choosing the **Window > Close** menu item in their respective windows.

You have completed this lab exercise.

Continue with Lesson 2: "Editing a Schematic for PCB" on page 2-1.

Lesson 2
Editing a Schematic for PCB

This lesson describes how to edit your schematic in Design Architect, and how to add design information needed by Board Station.
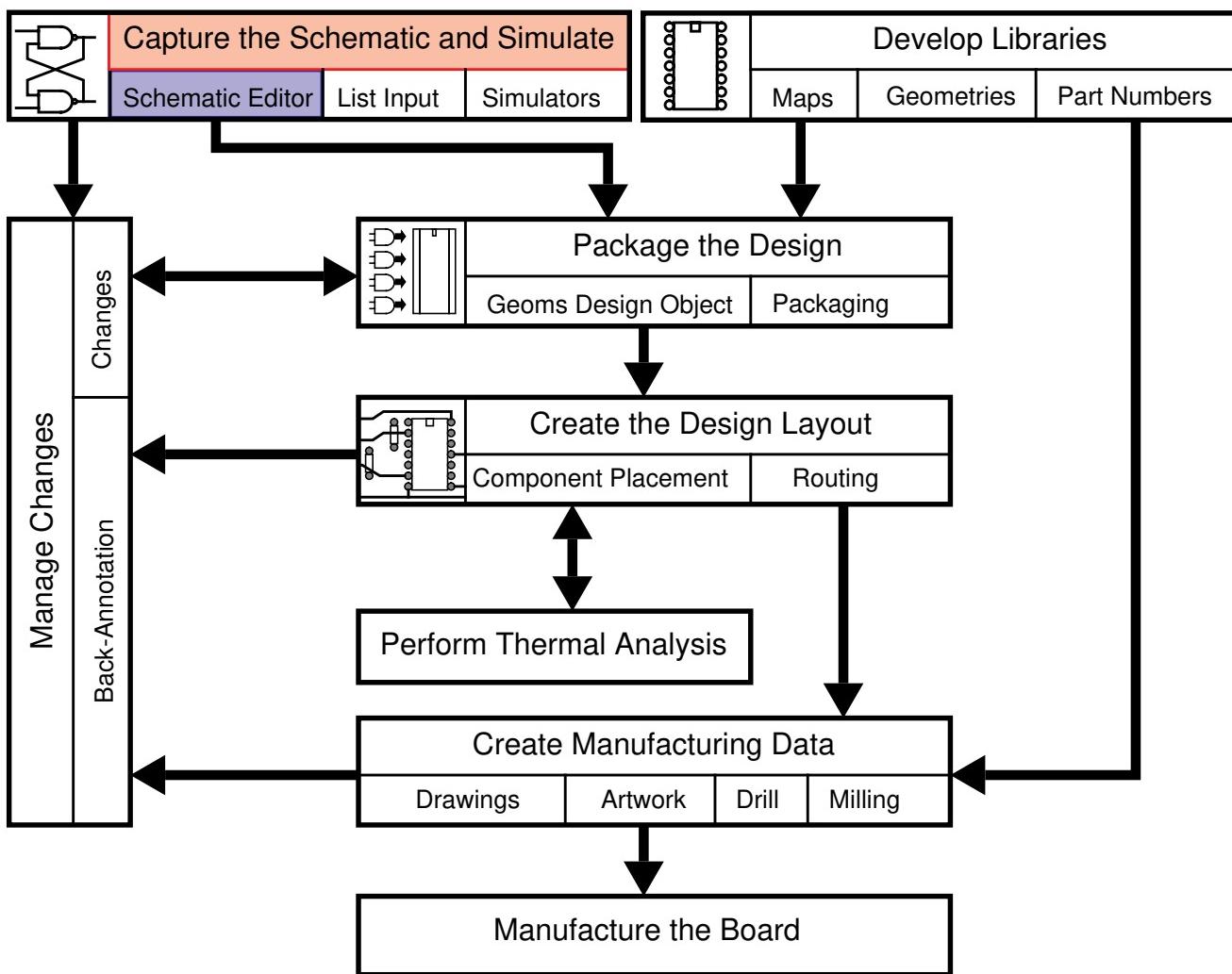
Figure 2-1. Board Process Flow Chart

Objectives

In the previous lesson you learned how to display your schematic in the Schematic Editor in Design Architect. You also learned how to view different areas of the sheet, and how to open the schematic sheet represented by a component.

This lesson discusses techniques for creating logical circuit design information which is the input for the physical circuit board design. The task of adding symbols and nets to a schematic, typically called design capture, is performed in Design Architect.

This lesson also explains the concept of Board Station properties. Properties are characteristics which are attached to objects on a schematic diagram to affect the behavior of the design.

After completing this lesson, you should be able to do the following:

- Activate library symbols and place them on a schematic sheet.

- Add nets to a design and identify which nets are connected.

- Explain why optional and user-defined properties are used.

- Describe six basic properties and their functions in PCB design data.

- Explain the purpose of symbol property switches.

- Describe why and how to add the Placement_region property to component instances.

- Check a schematic sheet in Design Architect.

Process for Preparing a Schematic for Board Station

To create a schematic in Design Architect for use in Board Station tools, use the following process. Each step of the process is discussed in detail in later sections of this lesson. In the lab exercise, you will follow this process.

1. Activate component symbols and place them on your schematic sheet.

2. Add nets (wires) to connect the symbols.

3. Add properties to convey design information to other applications, such as PACKAGE and LAYOUT.

4. Check the design and correct any errors.

5. Save the design.

MGC Libraries

Mentor Graphics component libraries contain software models of off-the-shelf devices. While in the Schematic Editor, you can place symbols from component libraries on your schematic sheet. Logic component libraries are named after the family of software models they contain. For example, *ls_lib* contains 74-series low-power Schottky component models, and *mil_ls_lib* contains the corresponding 54-series models. Component models in the libraries are named like the off-the-shelf devices they represent. For example, *74ls00* and *54ls243* are valid component model names.

Component libraries are a network resource which can be stored anywhere on the network. You access libraries by using soft prefixes defined for Mentor Graphics and custom libraries in a location map or environment variables. The two libraries mentioned above are accessed through the soft prefixes *$MGC_LSLIB* and *$MGC_MILLSLIB*, respectively. The complete pathnames associated with the soft prefixes are defined in a location map. User library locations may also be defined in a location map.

Contact your system administrator for more information about location maps and soft prefixes at your site.

Location maps are discussed in the section "Location Map" on page 1-3 of this workbook.

For additional information about location maps, refer to "Design Management with Location Maps" in the *Design Manager User's Manual*.

Placing Symbols on the Schematic

There are several methods you can use to place symbols on a schematic. Choosing a symbol from the Library Palette is the most common method and is described first.

The second method described here uses the Navigator to locate the desired symbol. Use this method to place symbols that are not in released libraries, or when you don't remember the pathname to a library and are not using a location map or environment variables to specify library locations.

The last methods, described in "Active Symbol" on page 2-7, are helpful when you want to place multiple instances of the same symbol on a schematic sheet.

Libraries Palette

Follow these steps to use the MGC Digital Libraries to place schematic symbols from a library on your schematic:

1. Click on the **Add_Route > Library** palette icon.

A list of Mentor Graphics libraries replaces the palette menu icons.

Mentor Graphics-supplied libraries can be stored anywhere on your network, and you should have either a location map or environmental variables that define pathnames to the libraries.

The names of all the Mentor Graphics libraries are displayed in the palette, whether or not you or your system manager have specified pathnames for each of them. You can also place names of user libraries in the libraries palette menu. You can choose components only from libraries for which the pathnames have been specified.

2. Click the Select mouse button on a library name, such as *gen_lib*, from the list in the Palette window.

The components included in the library are listed in the Palette window. Figure 2-2 shows the MGC Digital Libraries menu and the Gen_lib menu.
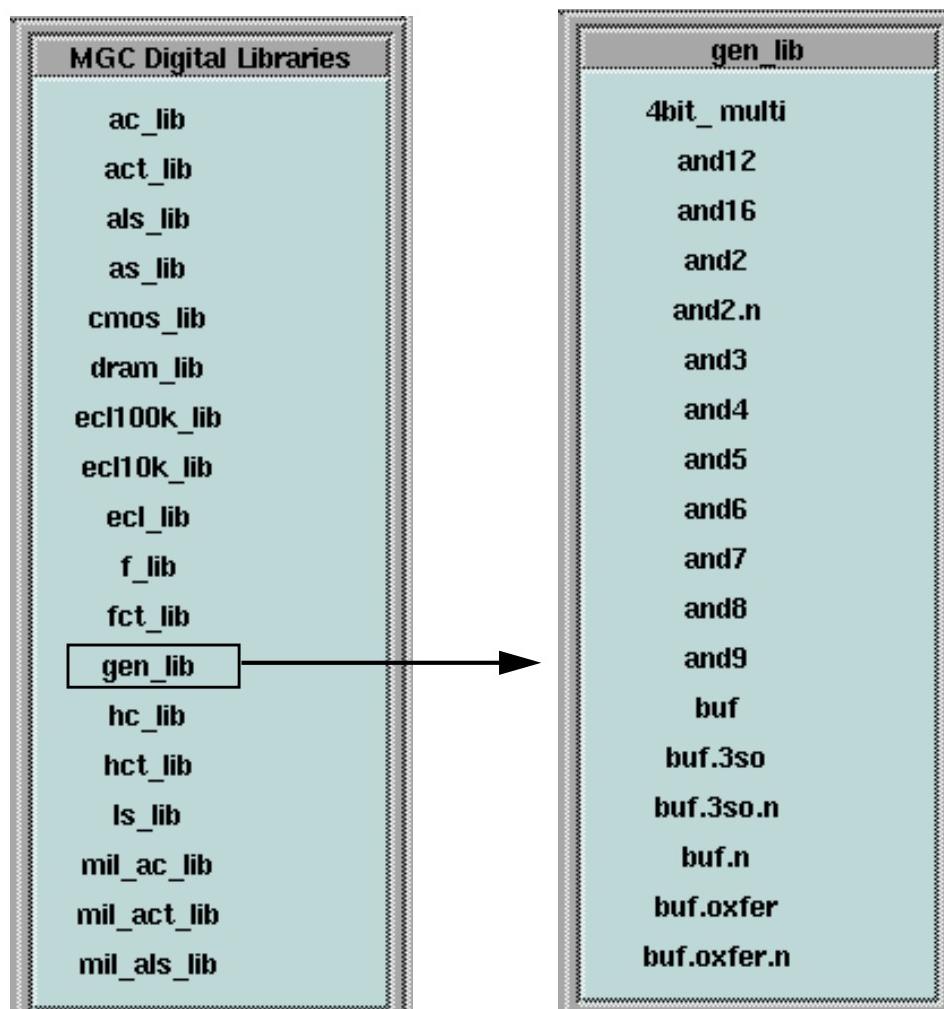
Figure 2-2. MGC Libraries Menu and Gen_lib Menu

3. Click on the component that you want to place on the sheet.

The component symbol you select is displayed in the Active Symbol window, and the Place Active Symbol prompt bar is displayed, prompting you for a location.

4. Move the cursor into the Edit window and notice that the ghost image of the active component symbol moves with the mouse.

5. Position the symbol image in the desired location and click the Select mouse button.

An instance of the symbol is placed on the sheet, and is selected. If you wish to move, copy, or delete the symbol, choose the appropriate palette icon or popup menu item.

Navigator

Follow these steps to use the navigator to find and place symbols on your schematic:

1. If the Library palette menu is displayed, choose the **Libraries > Display Schematic Palette** pulldown menu item.

2. Click on the **Add_Route Palette > Choose Symbol** icon.

The navigator is displayed.

3. Click the **Goto** button, shown at the left, on the Navigator.

A dialog box is displayed for you to enter a pathname, environment variable name, or soft prefix used in a location map, such as *$MGC_GENLIB*. The dialog box also displays a list of pathnames used previously; if you click on one of these pathnames, it is placed in the text entry box.

4. Choose or enter a pathname in the dialog box, and press **OK**.

The components in the selected library are displayed in the Navigator.

5. Choose a component, and click on **OK.**

    The Add Instance prompt bar is displayed, prompting you for a location.

6. Move the cursor in the Edit window to the location where you want the component symbol, then click the Select mouse button.

    An instance of the symbol is placed on the sheet, and is selected.

Active Symbol

The Active Symbol window, shown in Figure 2-3, is in the upper right hand corner of the Schematic Editor window. The component name and symbol name are shown at the bottom of the window.
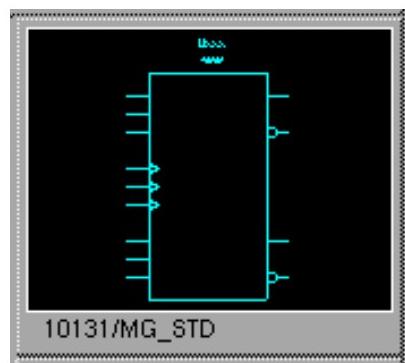
Figure 2-3. Active Symbol Window

To place an instance of the active symbol, either press the Place Active Symbol function key, or click the Select mouse button in the Active Symbol window. Both of these actions display the prompt bar.

Move the cursor (ghost image follows) to the desired location and click the Select mouse button to place the symbol instance.

Drawing Wires

A *net* is a signal (called a wire), or a set of signals (called a *bus*), that connects instances of symbols through multiple hierarchies of a design.
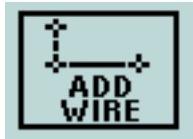
You create wires by clicking on the **Add_Route Palette > Add Wire** icon. A prompt bar displays, prompting you for a location. Move the cursor to the point where you want the wire to begin, such as an instance pin location, and create the wire as shown in Figure 2-4.
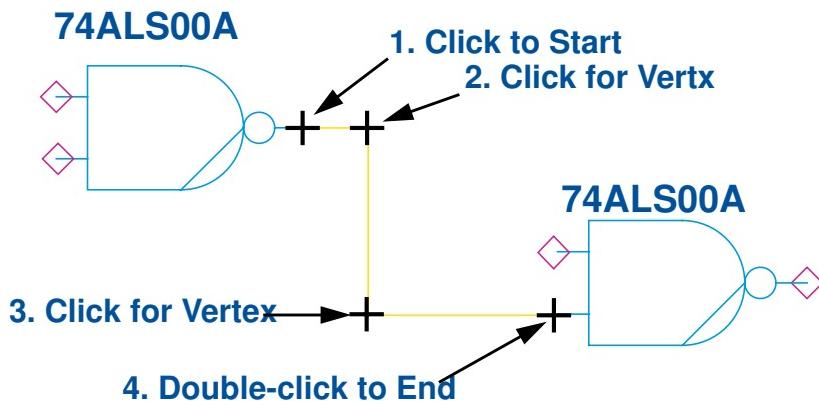
Figure 2-4. Creating Wires

You can create a wire with as many points (vertices) as desired. The Add Wire prompt bar is still displayed. Add more wires, or exit the Add Wire mode by clicking the **Cancel** button, as shown in Figure 2-5.

Figure 2-5. Exiting the Add Wire Mode

You create buses in the same way, except you use the **Add_Route Palette > Add Bus** icon.

Automatic Wire Routing

Design Architect has an automatic wire router. To activate it, choose the **Setup > Set Autoroute On** pulldown menu item. To use it, click on the desired beginning point and double-click on the desired end point for a wire or bus, as shown in Figure 2-6; the wire router orthogonally routes the wire around other nets, instances, and comment objects. If this causes a wire to cross another wire, those wires are not connected.

After you move a wire or bus, if autoroute is on, Design Architect automatically calls the wire router. The original wire is replaced by a new orthogonally routed wire that avoids other objects.

If the autorouter is not turned on, you can draw a wire or net as shown in Figure 2-6, then click on the **Palette > Route Selected** icon. The wire is routed as described above.
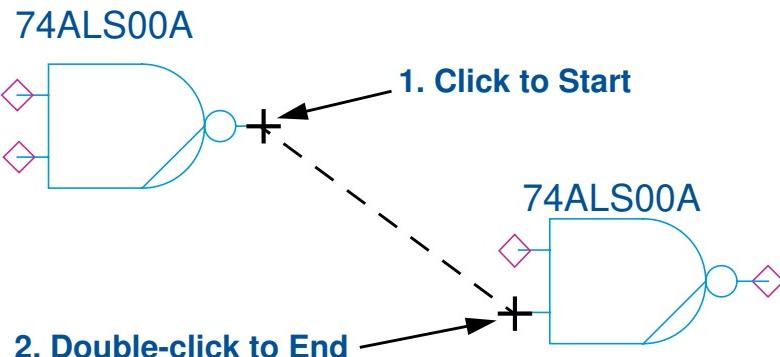
Figure 2-6. Using the Automatic Wire Router

Naming Wires

Although Design Architect assigns an internal system name to each net (a handle name like N$3), you can assign names that have more meaning. External nets, the nets connected to ports, must be named to match the pins on the associated symbol. You can also assign meaningful names to internal nets to make it easier for you to identify them in downstream applications.

All net names must be unique, and all buses must have a net name.

Two or more nets with identical names are treated as connected, even if they may not be graphically connected on the sheet. Two nets with identical names on different sheets in the same schematic are also treated as connected.

To add a net name, select a net vertex and click on the **Palette Text > Name Net** icon. The Add Property prompt bar, shown in Figure 2-7, is displayed for you to enter the Property Value (net name). When you move the cursor to the edit window, there is a rubber band-like connection between the selected net and the net name, which moves with the cursor. Move the cursor to the desired location for the net name, and click the Select mouse button.
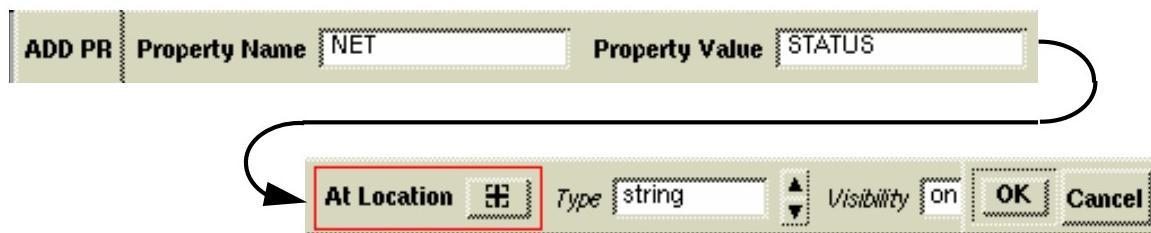
Figure 2-7. Add Property Prompt Bar

Net names are properties, which are discussed in "Properties" on page 2-12. You learn more about net names in the lab exercise.

Changing Names of Wires

All input wires and buses should begin with a **portin** or **offpag.in** component from the *gen_lib* component library. Similarly, all output wires and buses should terminate with a **portout** or **offpag.out** component. The portin and portout components assign the name *NET* to an unnamed wire or bus when attached. Wires or buses having the same name are seen as connected by downstream applications. You will need to change the names of these wires and buses so they each have a unique name.

To change the name, perform the following steps:

1. Press the Unselect All function key so nothing is selected.

2. Position the cursor on the name of the wire or bus.

3. Press the Change Text Value function key.

4. Enter a new name in the prompt bar, and **OK** the prompt bar. The Change Property Value prompt bar is shown in Figure 2-8.

Figure 2-8. Change Property Value Prompt Bar

Properties

Properties are fundamental to Mentor Graphics integrated design tools. Properties store information about your design such as the part number of a symbol, the name of a wire, or the number of a component pin. Properties assigned in one application provide information to other applications. For example, the Placement_region property controls the placement location of components in LAYOUT. You can add the Placement_region property in Design Architect, Design Viewpoint Editor (DVE), or the PCB PACKAGE application.

Some properties are optional, and some are required. For example, the PCB applications require each symbol instance in the schematic to have a Comp property, or equivalent, to identify the component represented by the symbol.

Some properties have a set of available values from which you can choose; other properties carry any value that you assign. Property values can include a period, comma, or slash character, but cannot include parentheses or quotations marks.

It is important for you to understand that properties carry information through the design process. Knowing how to find the information helps you interpret and analyze your design. Knowing when and how to add and modify properties allows you to control the design process.

For information about adding properties to a schematic, refer to "Design Capture Concepts" in the *Design Architect User's Manual.* Refer to "Properties" in the *PCB Products Design Reference Manual* for information about Board Station properties.

The properties discussed in the next subsections are shown in Figure 2-9.

Pin_no (Physical pin)   **Ref** (Reference designator)

Pin (Logical pin)

1   **in_1**

U24   **Out**   3

2   **in_2**

Inst (Instance) ——————▶ **I$123**   **Part_no** (Part number)

Comp (Component) ——▶ **5438**

SN54LS26W-10
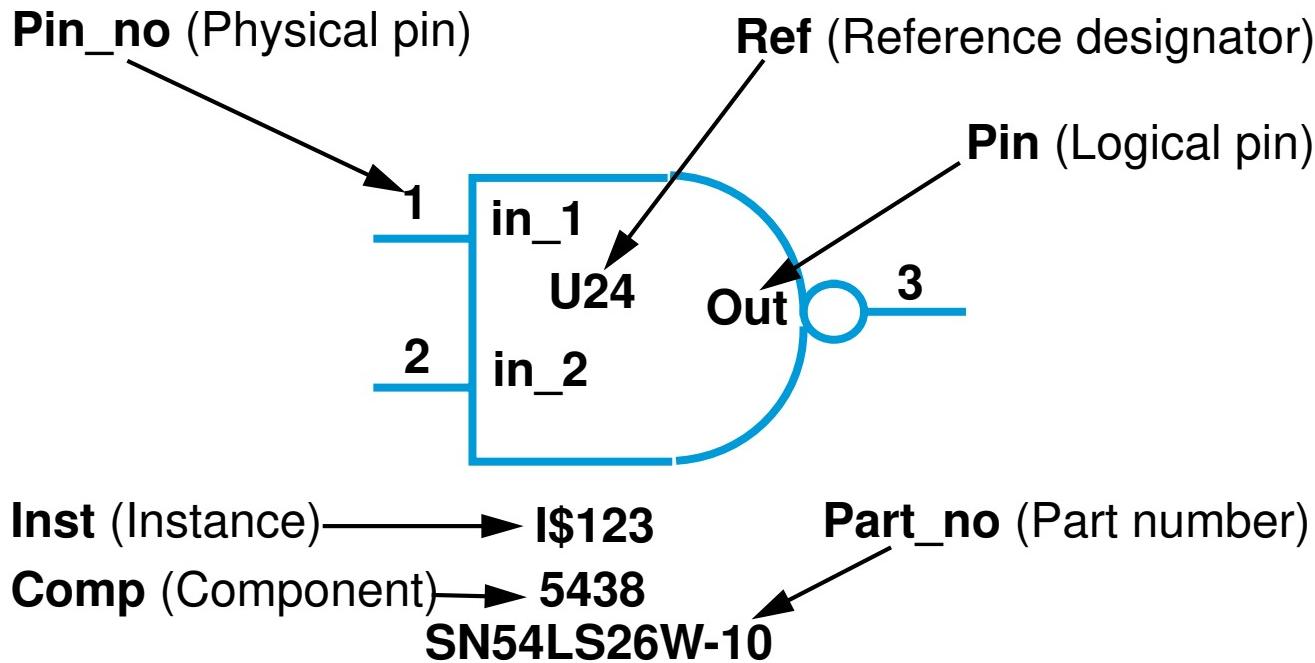
Figure 2-9. Symbol Properties

The Inst Property

The **Inst** property identifies an instance of a symbol on a schematic.
Each gate must have a unique Inst property value so that the tool can
identify all individual gates. For example, the symbol instance in
Figure 2-10 might appear in a design pathname as *I$11/I$22/I$123*.

Assigning an Inst property is one way to give an instance a meaningful
name that you will recognize. If you assigned the Inst property to the
instance with a property value of *area_a*, as shown in Figure 2-10,
then the design pathname to the instance would be *I$11/I$22/area_a*.

area_a
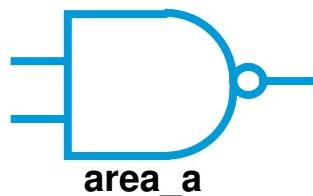
Figure 2-10. Inst Property

The instance handle assigned by the Design Architect Schematic Editor is used as the value of the Inst property if a user-supplied value does not exist.

The Comp Property

The **Comp** property specifies to the Package application how schematic symbols are assigned to physical parts. You assign this property to the symbol body during symbol creation in Design Architect. Each symbol on the schematic must have a Comp property, or a property defined as *equivalent* in the *pkgconf* design object (discussed in *Board Station for New Users Training Series*, Module 4: "Packaging the Design for Layout").

The Comp property value on a symbol must match at least one of the Comp property values in a catalog file in order to package the symbol. Comp property values that start with the letters *conn* and *edge* are reserved for connectors. The Comp property value shown in Figure 2-11 is *5438*.

7438
5438
74LS38
54LS38
74S38
54S38

5438
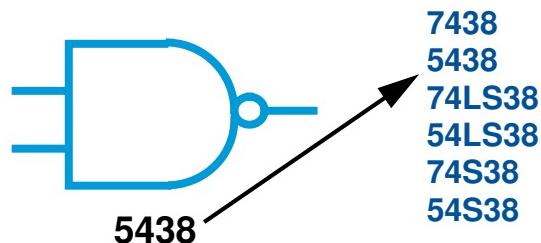
Figure 2-11. Comp Property

The Pin Property

The **Pin** property identifies the name of a logical pin on a symbol. This required property is assigned to each pin during symbol creation in Design Architect. The Pin property value is used to match with entries in the PCB mapping file and to support gate swapping. The Pin property values shown in Figure 2-12 are *in_1*, *in_2*, and *Out*.
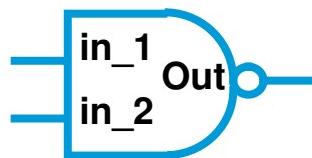
Figure 2-12. Pin Property

The Pin_no Property

The **Pin_no** property, illustrated in Figure 2-13, specifies the physical pin number to which the symbol's logical pin is assigned. This property is also assigned to each pin in Mentor Graphics libraries during symbol creation in Design Architect. If you create your own symbols, you must assign the Pin_no property to each symbol pin before checking the symbol in Design Architect.
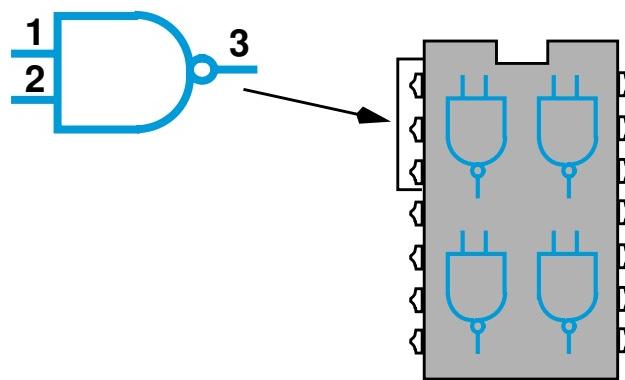
Figure 2-13. Pin_no Property

The Ref Property

The **Ref** property, shown in Figure 2-14, determines the reference designator. The Build program in PACKAGE uses this property to associate logical components with physical component packages. If you do not assign this property on the schematic, PACKAGE automatically assigns the reference designator to symbol instances.
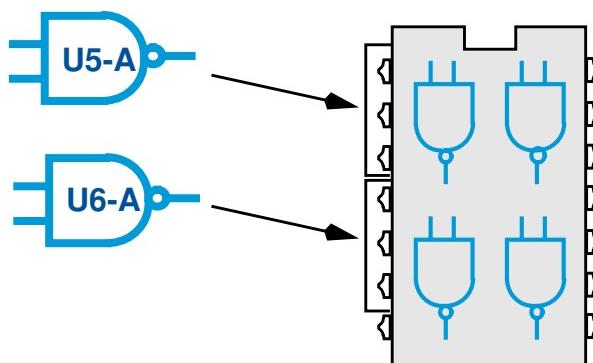
Figure 2-14. Ref Property

You can specify an incomplete reference designator on the schematic by using the wildcard character (?) at the end of the designator. For example, you could use the reference designator *Q?* for a transistor. When Build encounters the reference designator with a *?* at the end, Build replaces the *?* portion of the reference designator with a number starting from the current highest reference number and continuing from there.

Unless you specify otherwise, PACKAGE assigns reference designators beginning with *U1* (*J1* for recognized connectors).

Where multiple instances of a symbol on the schematic are assigned to a component, PACKAGE adds a symbol tag (such as *RPI-1*, *RPI-2*, ...) at the end of each reference designator. You can also add this symbol tag to the Ref property on the schematic, and it will be understood by PACKAGE.

The Part_no Property

The **Part_no** property, shown in Figure 2-15, specifies a part number, typically a stock number, for a given schematic symbol instance.

If the Part_no property exists on the symbol instance, PACKAGE matches that property value to the part number value in the catalogs to obtain mapping and geometry information.

SN54LS26W-10

Figure 2-15. Part_no Property

If the Part_no property is not present on a symbol instance in the schematic, PACKAGE searches the loaded catalog design objects for the correct part number to use for that symbol, and assigns that part number as the value of the Part_no property for the instance, if found.

Symbol Property Switches

When you add properties to symbol bodies and symbol pins during symbol creation, you also set *property switches*. These switches control the visibility and operations allowed on each property value on an instance of the symbol on a schematic sheet.

The *visibility* switch controls whether you can see the property text on a symbol instance on a schematic. The value may be either **Visible** or **Hidden**. Hidden property values are recognized by Design Architect, even though you cannot see them on the schematic.

The *stability* switch controls whether the property value can be changed on an instance of the symbol.

The stability switch only controls edits in Design Architect.

In PCB applications, properties cannot be deleted from a source view, but most properties can be added, changed, or deleted through back annotation in a PCB database.

Table 2-1 shows which of the four stability switches allow a property value to be changed on a symbol instance and deleted from an instance in Design Architect.

Table 2-1. Symbol Property Switches

Switch Setting	Change Value on Instance?	Delete from Instance?
Fixed	No	No
Protected	Only when first placed on the schematic	No
Variable	Yes	Yes
Nonremovable	Yes	No

The Pin and Pintype properties are Fixed by default. The Comp property is Fixed on most symbols; however, on generic symbols such as diodes, the Comp property is Protected, and you assign the device type as the Comp property value when you place an instance of the symbol on a schematic.

Most properties are Variable and Visible by default, although you can change the switches, as well as the value, when you add a property.

Use the Nonremovable switch setting for properties that are required by downstream applications, even though the property values may be changed.

Defining a Placement Region

You can control where components are placed on a board by assigning the Placement_region property to component instances. For example, if you want to group certain components in a particular physical area of the board, you can add the Placement_region property to those components. In LAYOUT, the property value is matched with names of areas you define on a circuit board. Examples of Placement_region property values are *analog* and *v15*.

This property is useful when you are creating a board with split power planes. You assign the Placement_region property to all components having the same voltage requirement, then create a placement area of the same name in LIBRARIAN.

You can add this property in Design Architect, PACKAGE, or LAYOUT. When you create the board geometry in LIBRARIAN, you define and name a physical area of the board. The name of the area must be the same as the Placement_region property value on the components in the dialog box. The auto placement feature in LAYOUT gives preference to components having the Placement_region property.

Perform the following steps to add the Placement_region property:

1. Select one or more components.

2. Choose the **Properties > Component Properties > Placement_region** popup menu item, as shown in Figure 2-16.

   The Add Placement_region dialog box is displayed.
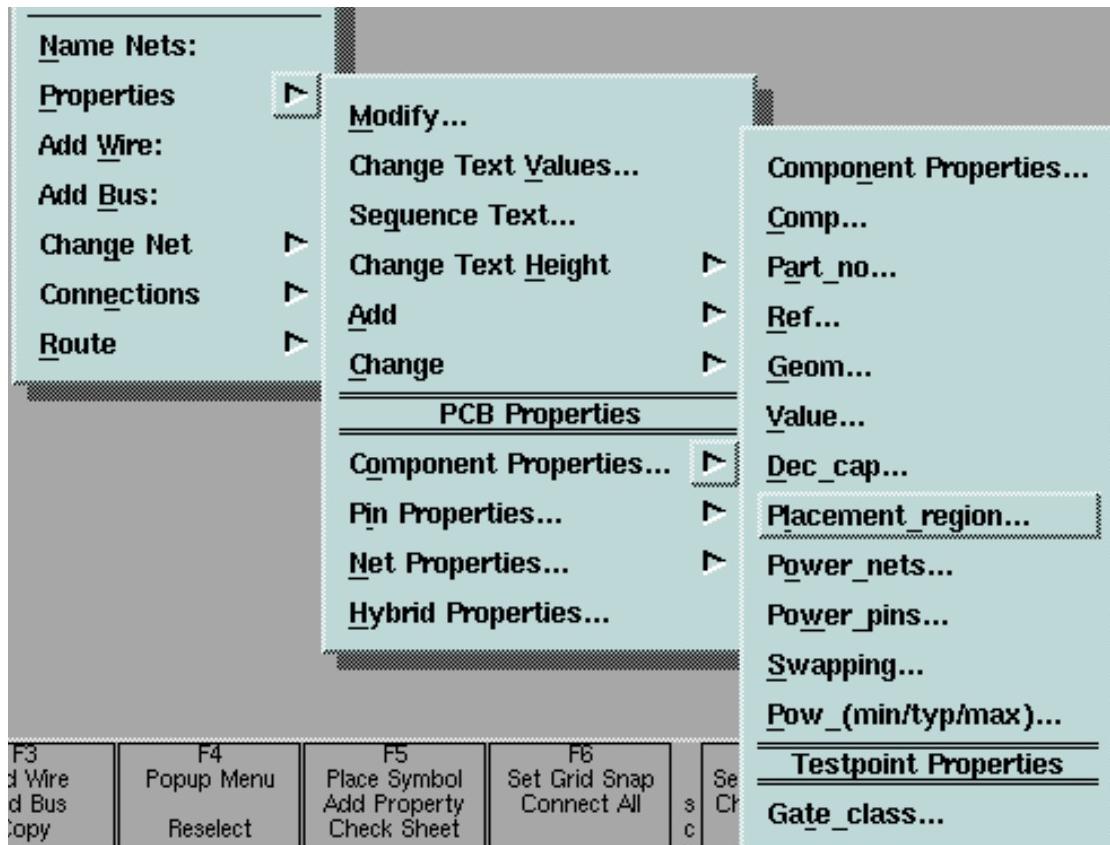
Figure 2-16. Properties > PCB Properties Submenu

3. Enter the name of the placement region as the property value, click on the **Visibility** button as needed to specify whether the property text should be visible or hidden on the schematic, then **OK** the dialog box.

4. Click the Select mouse button at the desired location for the property text if you specified in the dialog box that it is visible.

You can verify the value of the Placement_region property by leaving the components selected and choosing **Report > Object > Selected**. The requested report lists property names and values for the selected components.

You add this property to some components in the lab exercise.

Extracting Information from the Design

Every design object has an associated *handle*, which is a unique system-assigned identifier. Examples of handles are *I$23* for an instance, and *N$13* for a net. You request a report on selected design objects by choosing the **Report > Object > Selected > All** menu item. In Figure 2-17, notice that all objects are identified by their handles:

- I$3966 identifies the instance.

- Block property text (DATA_IO) is T$22848.

- Inst property text (also DATA_IO) is T$22849.

- Pin P$22850 is attached to vertex V$7411 of net N$3851.

- Pin property text (A(31:0) is T$22851.

- Pin P$22852 is attached to vertex V$7415 of net N$1410.

```
Report#1  sig_az

Reporting: Instance, Net, Pin, Property, Comment, Frame,
           Attached pin, Attached property,

Instance   Name
I$2118     /user/jeanf/training/board84nwp/mod2/sig_az/CONTROL
           _PALS/CONTROL_PALS

           Property   Name              Value             Type
           T$17124    INST              CONTROL_PALS      string

           T$9334     block             CONTROL_PALS      string

    Pins:      Name       Attached Vertex      of Net
    P$21702    A(31:0)            V$9338             N$3851
           Property   Name              Value             Type
           T$21704    pintype           IN                string
```

Figure 2-17. Report on Selected Instance

You can find most information simply by reading a report. Clicking on an object handle in a Report window selects that object on the design sheet. This is especially helpful when you request a report on many objects. For example, if you wanted to use the report in Figure 2-17 to locate pin P$22850, you would perform the following steps:

1. Activate the Schematic window.

2. Press the Unselect All function key.

3. Click the Select mouse button on the pin handle (P$22850) in the Report window.

4. Choose the **View > View Selected** pulldown menu item.

The pin you selected in the Report window will be selected and centered in the view of the sheet. You can also select multiple handles in a Report window to select those objects in your design.

Using System Functions

Design Architect and the Common User Interface provide many system functions to help you gather information about your design or about the design environment. For example, you can ask for attributes of a property owned by an object, or ask for the current grid settings.

System functions are usually not available through menus. You simply place the cursor in the Edit window and begin typing the function. A popup command line appears, as shown in Figure 2-18.

Schematic#1 $get_instance_handles()
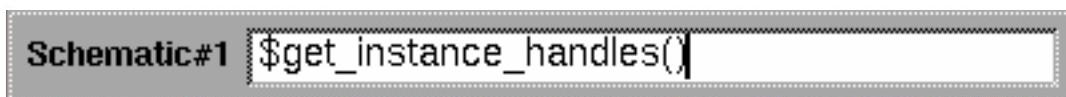
Figure 2-18. Popup Command Line

Returned values from system functions are displayed in the transcript. Choose **MGC > Transcript > Show Transcript** to display a transcript window.

Most system functions returning information about an object require an object handle, which you get from a report on the object. Another method of getting handles is to select one or more objects, execute the $get_select_handles() function, then use the returned values as input to other functions.

If you use a function within a function, be aware that some functions return strings and others return vectors. Refer to the *Design Architect Reference Manual* for details about functions and their requirements and returned values.

This example shows how to specify the first element of a vector as an input string. To find the names of all properties owned by a component instance, select the instance and enter the following in a popup command line:

> **$get_property_names($get_select_handles()[0])**

In this example, $get_property_names() requires a string, but $get_select_handles() returns a vector. The *[0]* specifies the first element in the returned vector. The properties owned by the selected instance are listed in the transcript.

Selecting Objects by Property

You can select objects that you know have a particular property or property value. For example, you might want to select all instances having a particular Placement_region property value. To select all instances having the Placement_region property with a value of *analog*, perform the following steps:

1. Choose **Select > By Property > Name-Value-Type** from any of the Schematic window popup menus.

2. Enter the following in the Select By Property dialog box, and click the **OK** button.

> Name: **Placement_region**
> Value: **analog**

You can enter multiple property names and values in the dialog box. To select objects that own any of the properties you specify, click on the **Union** button on the dialog box. To select only the objects that own all of the specified properties, click on the **Intersection** button in the Select By Property dialog box.

If you know the property name and/or value and want to select the property text, instead of the owning object, you can click on the **Text** button in the dialog box.

If you want to select property text, but don't necessarily know the property name or value, you can set the select filter by clicking the Select mouse button on the **Set Select Filter** button in the palette menu. In the dialog box, click on **Properties,** and **OK** the dialog box. After setting the selection filter, you can select a property by clicking on the text, or you can select all properties within an area by defining the selection area using the mouse.

If you need to know a property name or value, select all properties within an area, then choose the **Report > Object > Selected > All**. Each property in the selection area is listed by handle, with the property name, value, location, and switch settings.

The select filter is set for all windows of the same type (one filter for all Schematic windows, another filter for all Symbol windows), and the settings remain until you change them or close the Session window.

Checking and Saving the Design

Your design must pass a default set of Mentor Graphics checks before you can use it in a downstream application. A default check report will list errors and warnings. If any errors are found in a sheet, they must be corrected for the sheet to pass the checks. Examples of errors are invalid pin or net names, and incorrect number of pins.

Warnings in a check report indicate possible problems in the design, such as unconnected pins. Warnings do not stop a design from passing checks.

If the design fails the checks, other applications will issue warnings that problems may be found at a later time. After checking your design sheets, you need to save each sheet.

Checking a Single Sheet

You need to check each sheet in the design. Choose the **Check > Sheet > With Defaults** pulldown menu item to check the sheet you are currently editing. The following list includes examples of the checks performed on a sheet:

- Symbol instances on the sheet are checked for valid pins and pin names, valid syntax of property values, and references to current symbol models.

- Special instances, such as bus rippers, are checked for proper number of pins and valid property value syntax.

- Nets are checked for valid net names, and proper range specifications for buses.

- Frames are checked for valid frame expression syntax, and for other objects that overlap the frame border.

There are optional checks that you can specify, and you also have the ability to write and invoke your own sheet checks. Required and optional checks are listed in Appendix A: "DA Design Checks" in the *Design Architect User's Manual*. Refer to "Design Error Checking" in

the *Design Architect User's Manual* for information about setting up the Check command and for writing user-defined checks.

A check report is shown in Figure 2-19. Notice that required checks are marked in the report, and warnings and errors, if any, list object handles. To check the unconnected pin listed in the report, select it on the sheet by clicking the Select mouse button on the *P$3317* in the Report window.

```
Check#1  sig_az:Sheet

Check Sheet "sig_az/schematic/sheet1"
Check SymbolPins --------    0 errors   0 warnings   (MGC-required)
Check Overlap -----------    0 errors   0 warnings
Check NotDots -----------    0 errors   0 warnings
Check Closedots ---------    0 errors   0 warnings
Check Dangle ------------    0 errors   1 warnings
  Warning:  Unconnected pins on
            Instance(s)  at Pins(s):
            I$2329       P$11223     P$11343
            I$2118       P$9535


Check INIT Properties ---    0 errors   0 warnings
Check Owner -------------    0 errors   0 warnings
Check Instance ----------    0 errors   0 warnings   (MGC-required)
Check Special -----------    0 errors   0 warnings   (MGC-required)
Check Net ---------------    0 errors   0 warnings   (MGC-required)
Check Frame -------------    0 errors   0 warnings   (MGC-required)

  "sig_az/schematic/sheet1" passed check :   0 Errors, 1 Warnings
```

Figure 2-19. Check Report

Checking the Schematic

Although you must check individual sheets in your design, you are not required to perform a schematic (multi-sheet) check before invoking other applications on your design.

However, should you choose **Check > Schematic > With Defaults**, all sheets in the same level of hierarchy of the design are first checked as individual sheets, then multi-sheet checks are performed. Multi-sheet checks include checking for matching pins and ports, unique Inst property values within the schematic, matching on/off-page connectors, and shorted global nets. Schematic checking in Design Architect does not include hierarchical checking, other than matching symbol pins to net names.

Saving a Sheet

Any model (schematic, symbol, VHDL) must be registered with a component before it may be used in another application. When you choose the **File > Save Sheet** menu item, shown in Figure 2-20, you automatically register the sheet with the component specified when you opened the sheet.
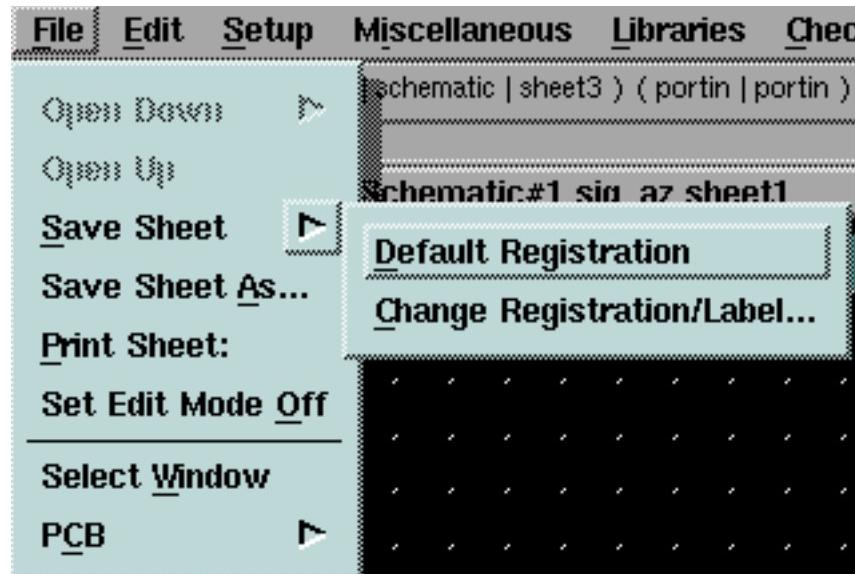

Figure 2-20. File > Save Sheet Menu

If you want to register your schematic sheet with other components, or with other interfaces within a component, you need more information about model registration, which is beyond the scope of this training series. "DA Model Registration" in the *Design Architect User's Manual* discusses concepts related to component structure and model registration.

Lab Exercise

In this lab exercise, you edit a schematic sheet by adding symbols and connections. You also add properties to the sheet to control the placement of components later in the board design process.

Upon completion of this lab exercise you should be able to:

- Activate symbols in a library and place the symbols on a schematic.

- Connect the symbols by adding nets.

- Add a Placement_region property to symbol instances.

- Check and save the changed schematic.

- Select objects by property.

Turn to Module 2—Lab 2: "Editing a Schematic for PCB".

Lab 2
Editing a Schematic for PCB

Introduction

In this lab exercise you complete a schematic sheet by adding instances of symbols and wire connections. You also add properties to the design to control placement of geometries later in the board design process. Finally, you check and save the schematic sheet.

Upon completion of this lab exercise, you should be able to:

- Activate symbols in a library and place the instances of the symbols on a sheet.

- Connect the symbol instances by adding wires.

- Add a Placement_region property to symbol instances.

- Check and save the changed schematic.

- Select objects by the properties they own.

- Use system functions to obtain design information.

Procedure

In this lab exercise you use Design Architect to complete a schematic sheet.

Preparation for Lab

You are going to open a different sheet in the same design using the same method as you did in the previous lab.

1. Invoke Design Manager from a shell, if it is not already displayed.

   $MGC_HOME/bin/dmgr

2. Invoke Design Architect by double-clicking the Select mouse button on the Design Architect icon in the Tools window.

   Design Architect opens in a new window.

3. Click the **Maximize** button in the upper right-hand corner of the Design Architect Session window.

   If you are working on a Sun workstation, drag a corner of the window to enlarge it.

4. Click on the **Palette > OPEN SHEET** icon.

   The Open Sheet dialog box is displayed.

5. In the Open Sheet dialog box, click on the **Navigator** button, then click on the **Goto** button to navigate to the lab data:

   your_path**/training/board_new/mod2**

6. Click the down arrow on the Navigator to explore the contents of **mod2**. Select the design object, **sig_az** by moving the cursor to the name and clicking the Select mouse button, then **OK** the Navigator. The pathname to the **sig_az** design object is automatically placed in the Component field of the Open Sheet dialog box.

7. Change the sheet name to **sheet2,** and **OK** the dialog box.

   Sheet2 of the schematic for the component **sig_az** appears in a Schematic Editor window.

8. Maximize the Schematic window and view the entire sheet using the View All stroke.

   Sheet2 is not complete; Figure 2-21 on page 2-31 shows the completed schematic diagram for you to use as a guide in this lab. The dotted lines enclose the symbols and wires you will add to the schematic.
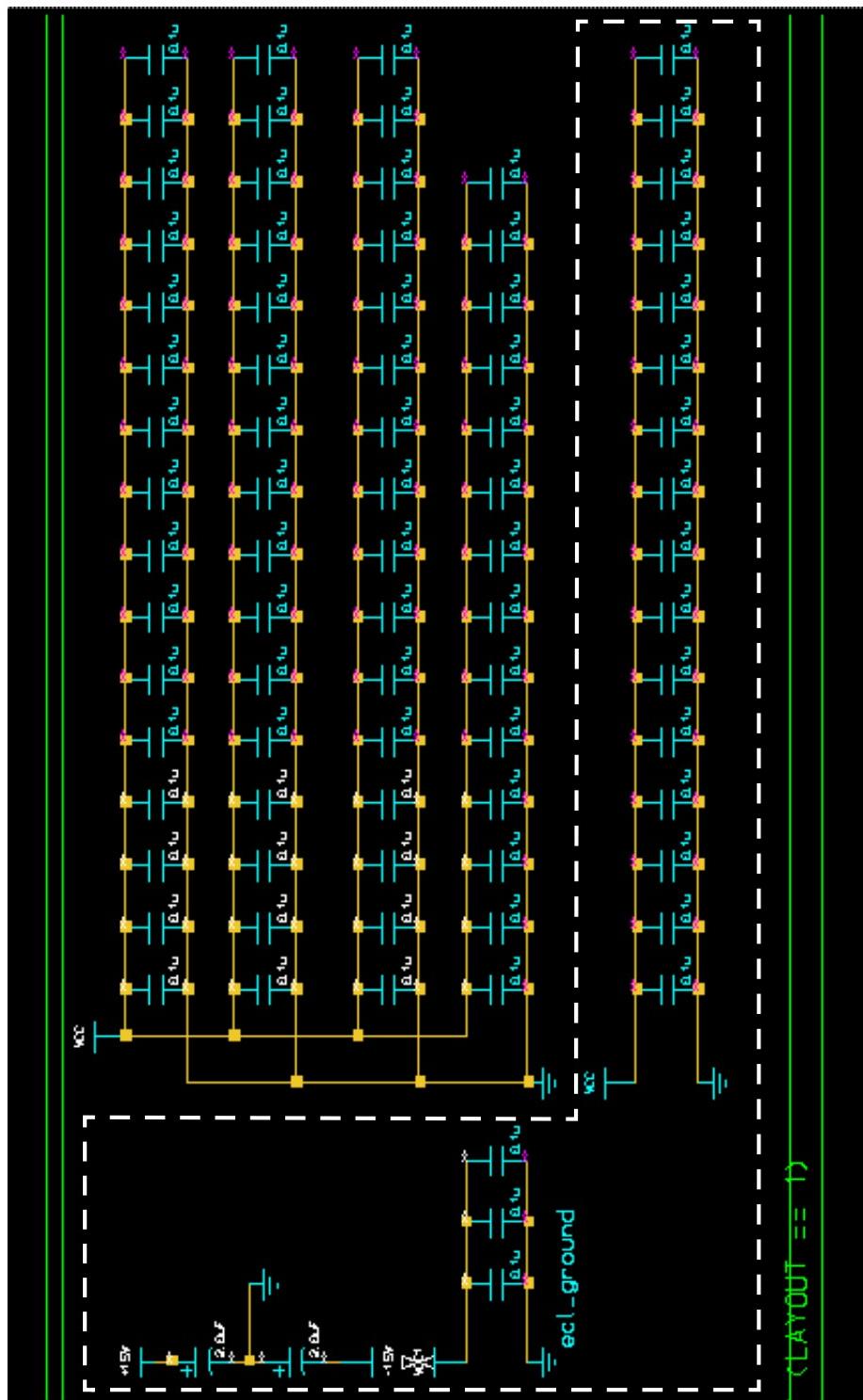
Figure 2-21. Complete Sheet2

Activating Library Symbols and Placing on Schematic

In this part of the lab exercise, you will choose some symbols from component libraries, and you will copy components already on the schematic.

1. Use the View Area stroke to display the area to the left of the existing circuitry.

   Figure 2-22 is a portion of the previous picture. The arrows indicate diagonally opposite corners of the viewing rectangle. If the pin grid is not visible, make the viewing area slightly smaller.
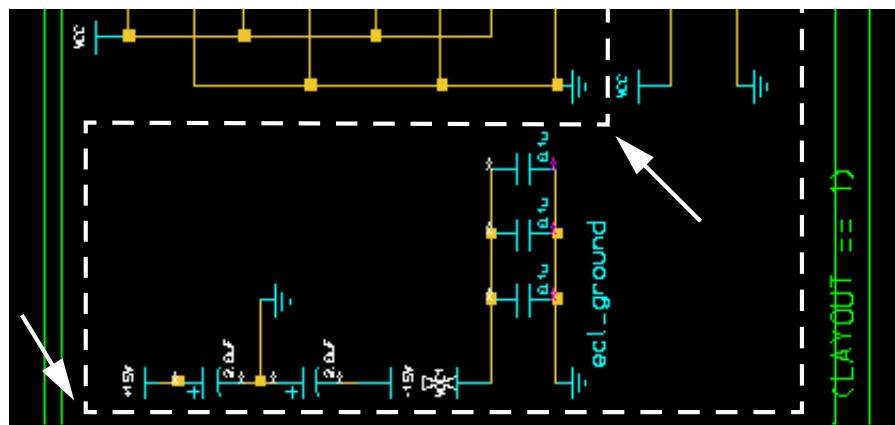

Figure 2-22. View Area
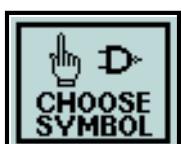
2. Click on the **Add_Route Palette > Choose Symbol** icon.

   This displays the Choose Symbol dialog box and navigator.

3. Click on the **GoTo** button on the navigator, and change the directory to:

   your_path/**training/board_new/mod2/gen_lib**

4. Scroll through the list of components, and choose **pwr_15** using the Select mouse button. There is no need to change any of the properties for this symbol. **OK** the dialog box.

   Use Figure 2-23 as a placement guide for adding symbols in the following steps.
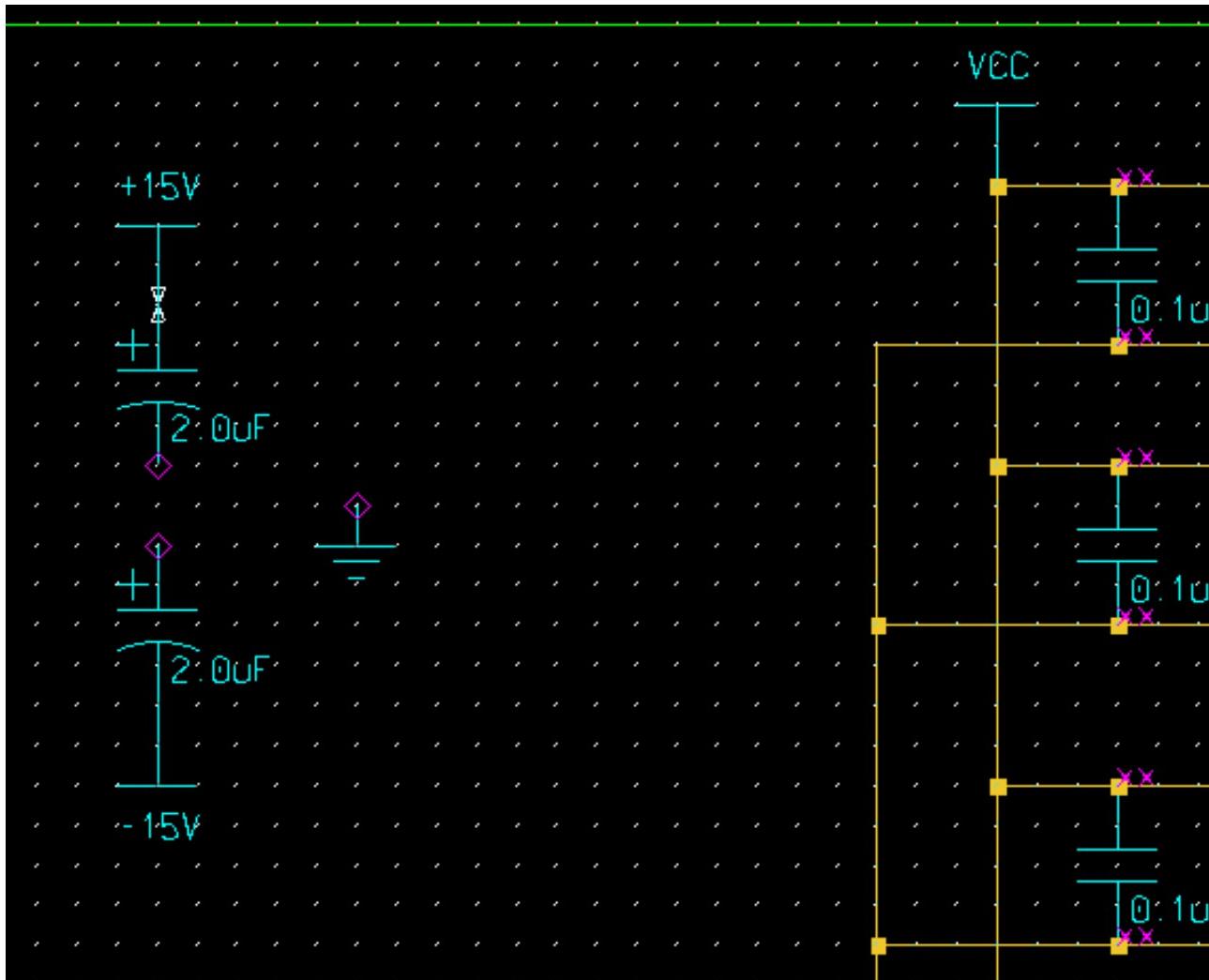
Figure 2-23. Placing Symbols on Sheet2

5. When the prompt bar appears, move the cursor into the Edit
    window to see a ghost image of the symbol. Move the cursor to
    drag the image to its location 21 grid points to the left and three grid
    points down from the **VCC** symbol instance.

6. Click the Select mouse button to place the **pwr_15** symbol.

    The prompt bar disappears when the symbol is placed. When
    objects are added to the schematic sheet, they are automatically
    selected.

7. Unselect the symbol instance by pressing the Unselect All function key.

8. Click the **Add_Route Palette > Choose Symbol** icon again.

    The Navigator still displays the list of *gen_lib* components.

9. Select and place the **pwr_-15** symbol 12 grid points below the **pwr_15** symbol, but do not unselect it.

10. Choose the **[Instance] Rotate/Flip > Flip > Vertical** popup menu item. If the placement is not correct, as shown in Figure 2-23, click on the **Palette > Move** button. When the Move prompt bar is displayed, move the ghost image of the component to its proper place and click the Select mouse button.

11. Unselect the symbol by pressing the Unselect All function key.

12. Click the **Add_Route Palette > Choose Symbol** icon. Using the navigator, move up one directory, then navigate to:

    your_path**/training/board_new/mod2/parts**

13. Click the Select mouse button on **cap.pol** in the list of symbols. Enter the changes in the Property Name and Property Value fields in the dialog box as shown in Figure 2-24, then **OK** the dialog box.
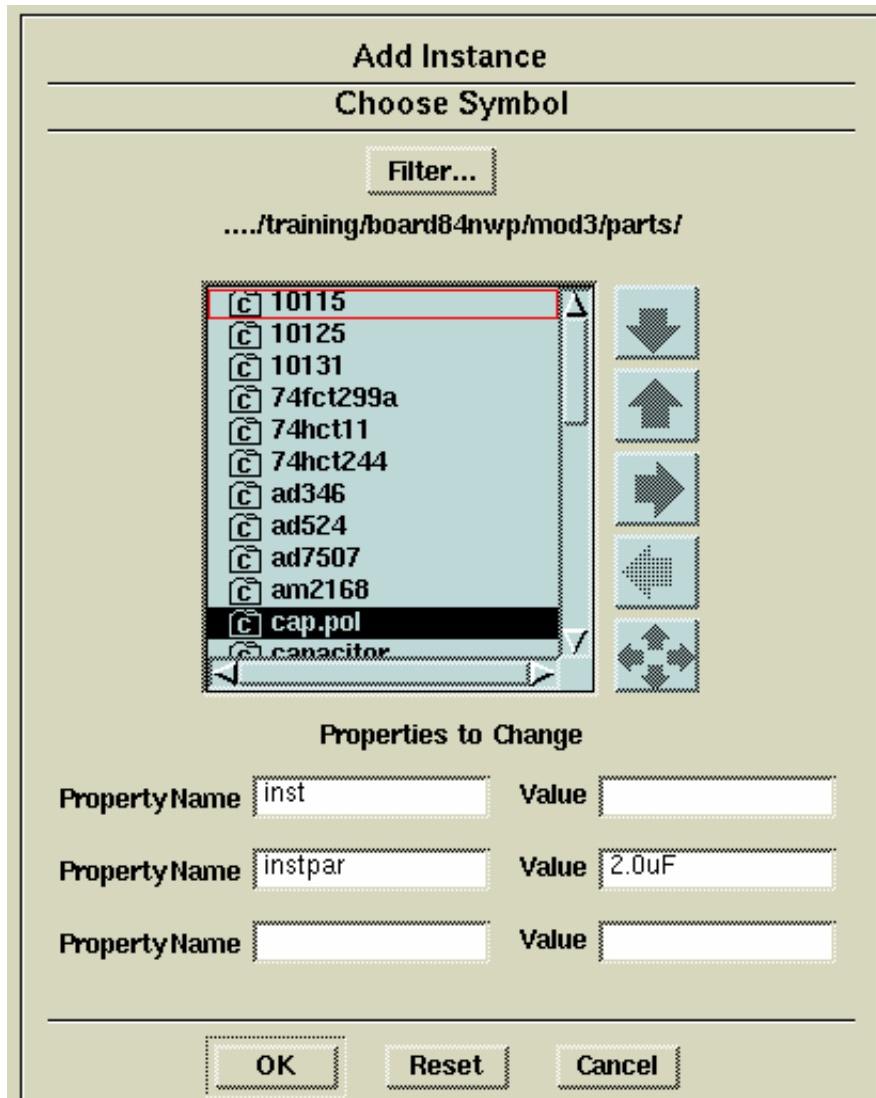
Figure 2-24. Add Instance Dialog Box

14. Place the capacitor so that its upper pin touches the lower pin on the **pwr_15** symbol, as shown in Figure 2-23.

15. Choose the **[Instance] Copy** popup menu item. When the Copy prompt bar appears, move the ghost image of the symbol so that its bottom pin touches the pin on the **pwr_-15** symbol. Click the Select mouse button to place the symbol.

not-dot

Design Architect places a *not-dot* (shown at left) where the two symbols touch. A not-dot is a circle with a diagonal slash through it (like an international "no" symbol), indicating there is no connection, even though there appears to be. When you move or copy a symbol instance so that there could be a connection, Design Architect places the not-dot to call your attention to it. When you initially place a symbol instance so that it touches another pin or a wire, that action is assumed to be intentional, so a connection is made.

16. Click the Select mouse button on the **pwr_-15** symbol to select it.

17. Make an electrical connection where the not-dot appears by choosing the **[Mixed Selection] Connections > Connect Selected** pulldown menu item.

    The not-dot disappears, so you know the connection is made.

18. Press the Unselect All function key, then click on the **Add_Route Palette > Choose Symbol** icon.

19. In the Choose Symbol dialog box, navigate back to **gen_lib**. Choose the **ground** symbol from the list of **gen_lib** components.

20. Drag the ghost image five grid points to the right of the capacitors and click the Select mouse button to place the symbol, as shown in Figure 2-23.

    The ground symbol is still the active symbol, and you need to place another one 11 grid points below the ground symbol that was originally on the schematic.

    Figure 2-25 shows symbol placement for the next steps.

21. Place an instance of the active symbol (ground) by clicking the Select mouse button in the Active Symbol window.

22. Move the cursor into the Edit window, drag the ghost image 11 grid points below the original ground symbol, and click the Select mouse button to place the symbol, as shown in Figure 2-25.
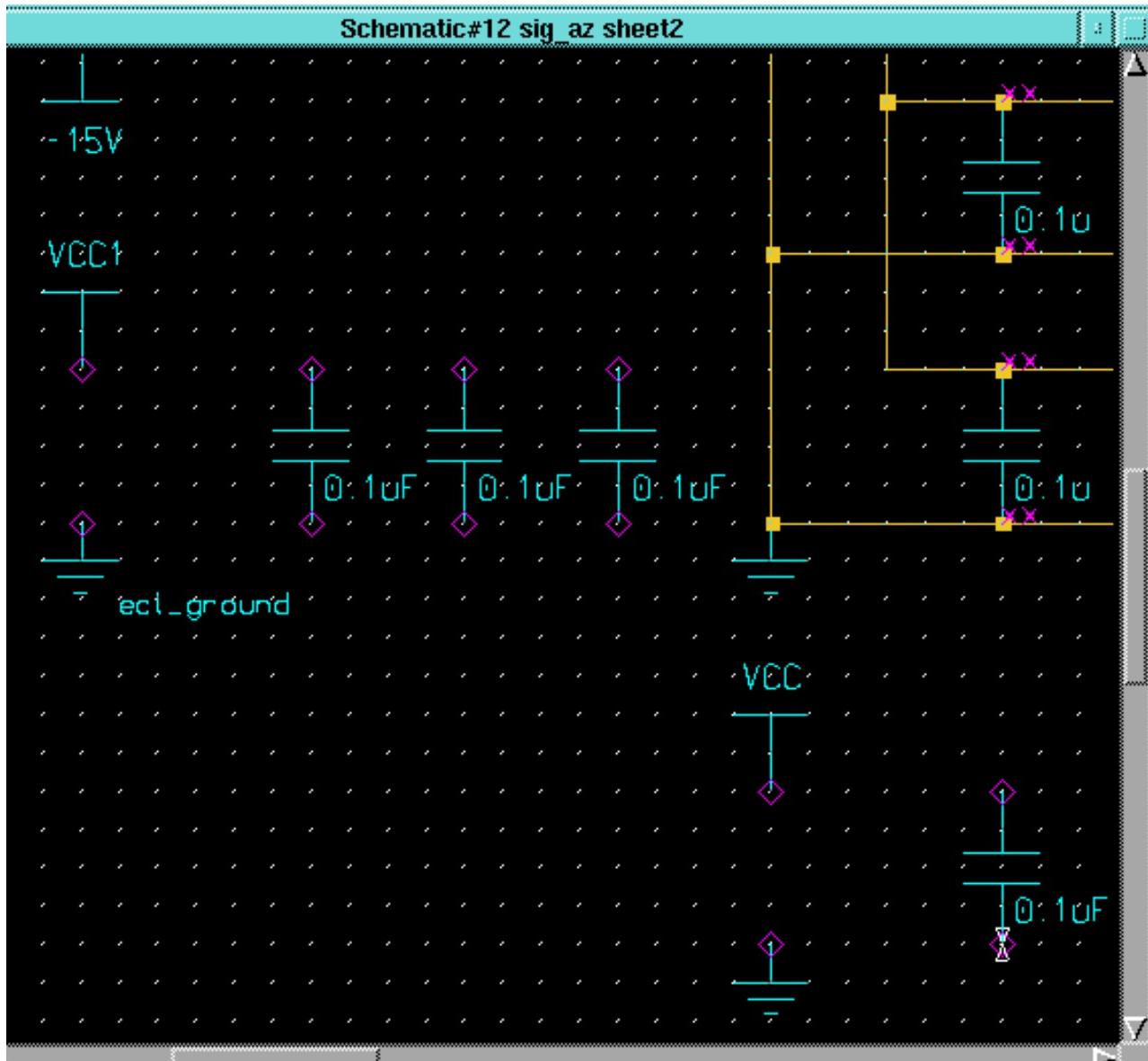
Figure 2-25. Placing Symbols

23. Place the **vcc**, **vcc1**, and **ground_ecl** symbols from **gen_lib** at the locations shown in Figure 2-25, using the same method as for previous symbols.

24. Unselect everything by pressing the Unselect All function key.

Now you need to place the remaining capacitors.

25. Click the **Choose Symbol** icon and navigate to **mod2/parts**. Choose **capacitor** from the parts list in the dialog box. Enter changes in the dialog box as shown in Figure 2-26.
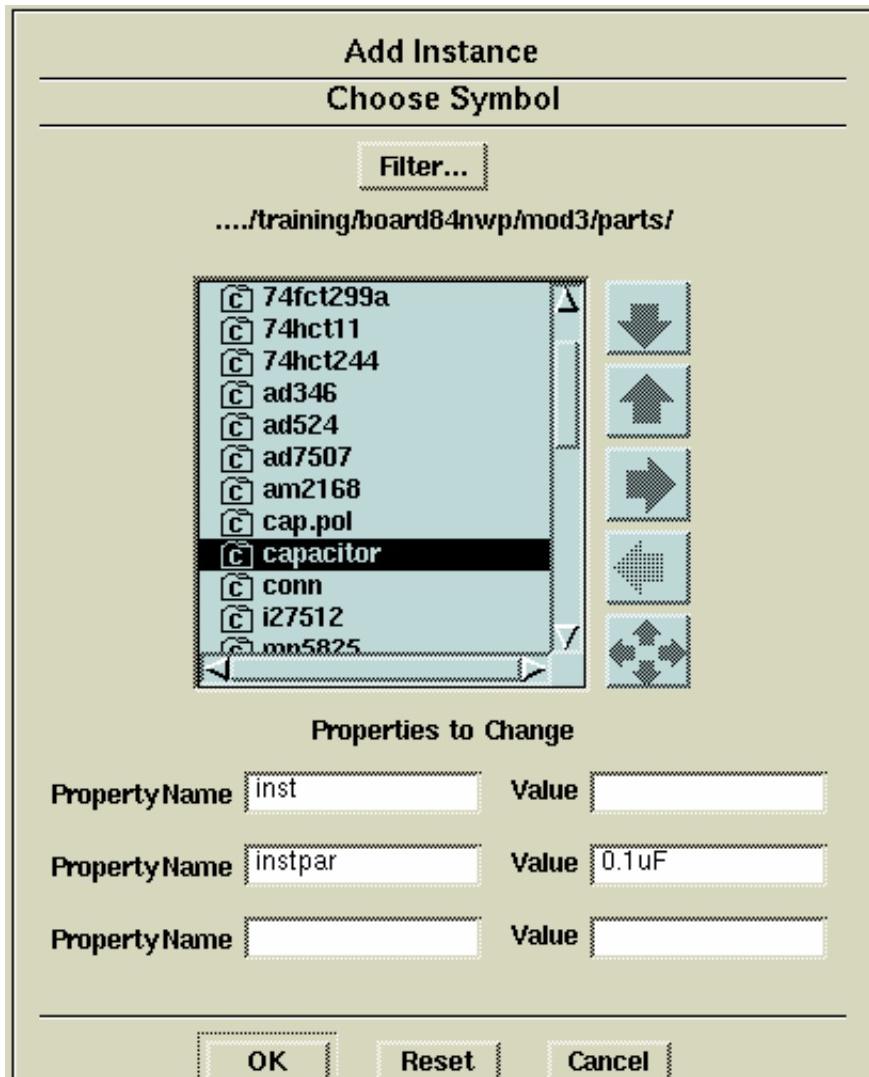

Figure 2-26. Add Instance Dialog Box for Capacitor

26. **OK** the dialog box, and place the symbol six grid points to the right of the **vcc1** and **ground_ecl** symbols, as shown in Figure 2-25.

Check the select count in the status line. The capacitor you just placed should be the only object selected.

27. Click on the **Palette > Copy** icon. When the Copy prompt bar appears, drag the image four grid points to the right of the first capacitor. Click the Select mouse button to place the copy.

Design Architect unselects the original and selects the copy.

28. Click on the **Palette > Copy** icon again, and place a third copy of the symbol four grid points to the right of the first two.

29. Press the Unselect All function key.

30. The capacitor is still the active symbol; click the Select mouse button in the Active Symbol window. Drag the ghost image of the capacitor six grid points to the right of the **vcc** and **ground** symbols, as shown in Figure 2-27. Click the Select mouse button to place the symbol.

Now you will place the row of capacitors across the bottom of the schematic.
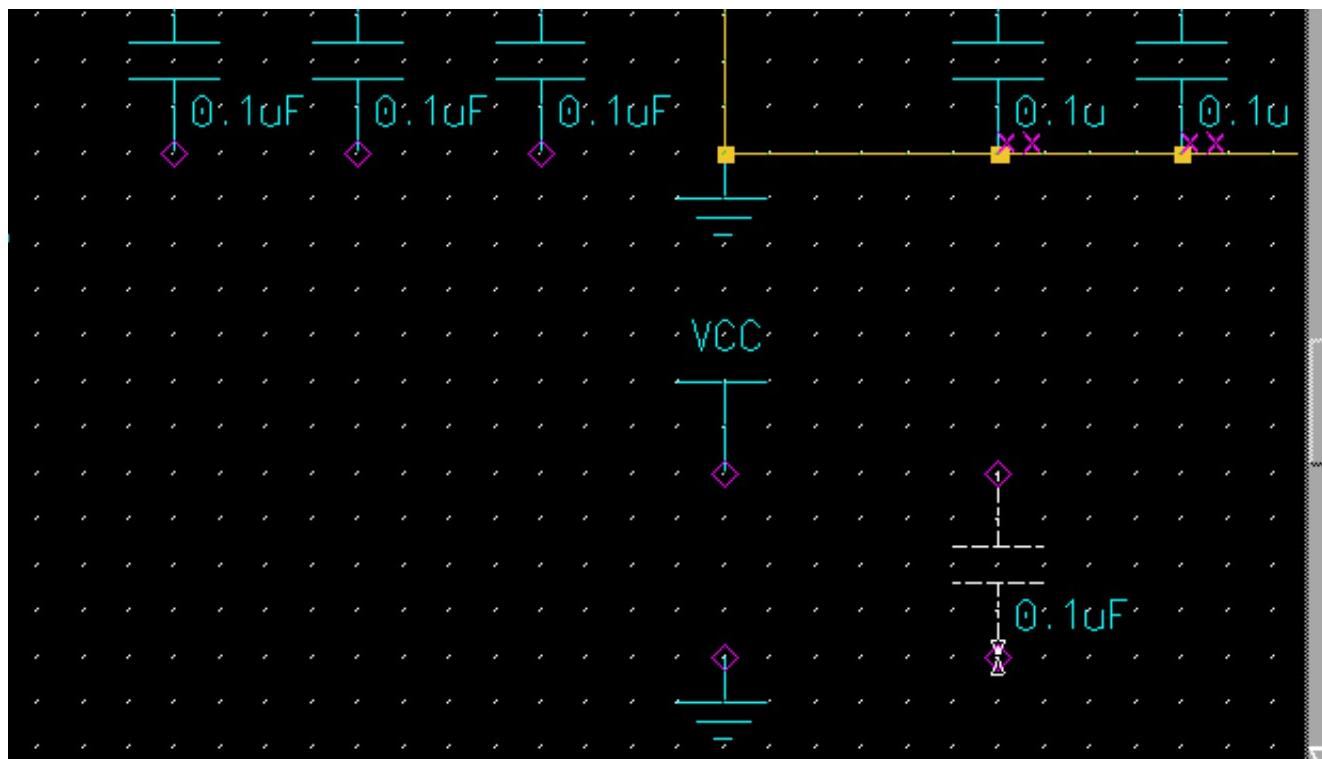
Figure 2-27. Starting the Bottom Row of Capacitors

31. Cancel the Select Area prompt bar.

32. Create the bottom row of capacitors by selecting the last capacitor you copied in the previous step, choosing the **Copy > Multiple** popup menu item, shown in Figure 2-28.
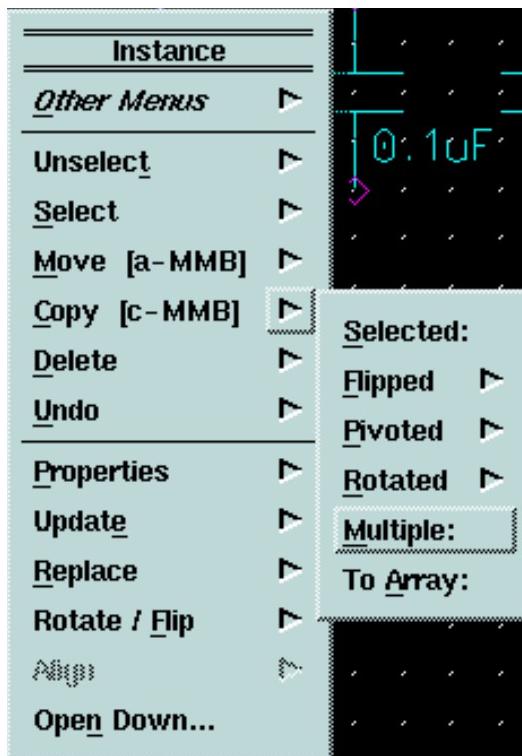

Figure 2-28. Copy Menu

Complete the prompt bar as shown in Figure 2-29.

Figure 2-29. Copy Multiple Prompt Bar

33. OK the prompt bar, drag the ghost image four grid points to the right of the first capacitor, as shown by the arrow in Figure 2-30. Click the Select mouse button. Press the Unselect All function key.
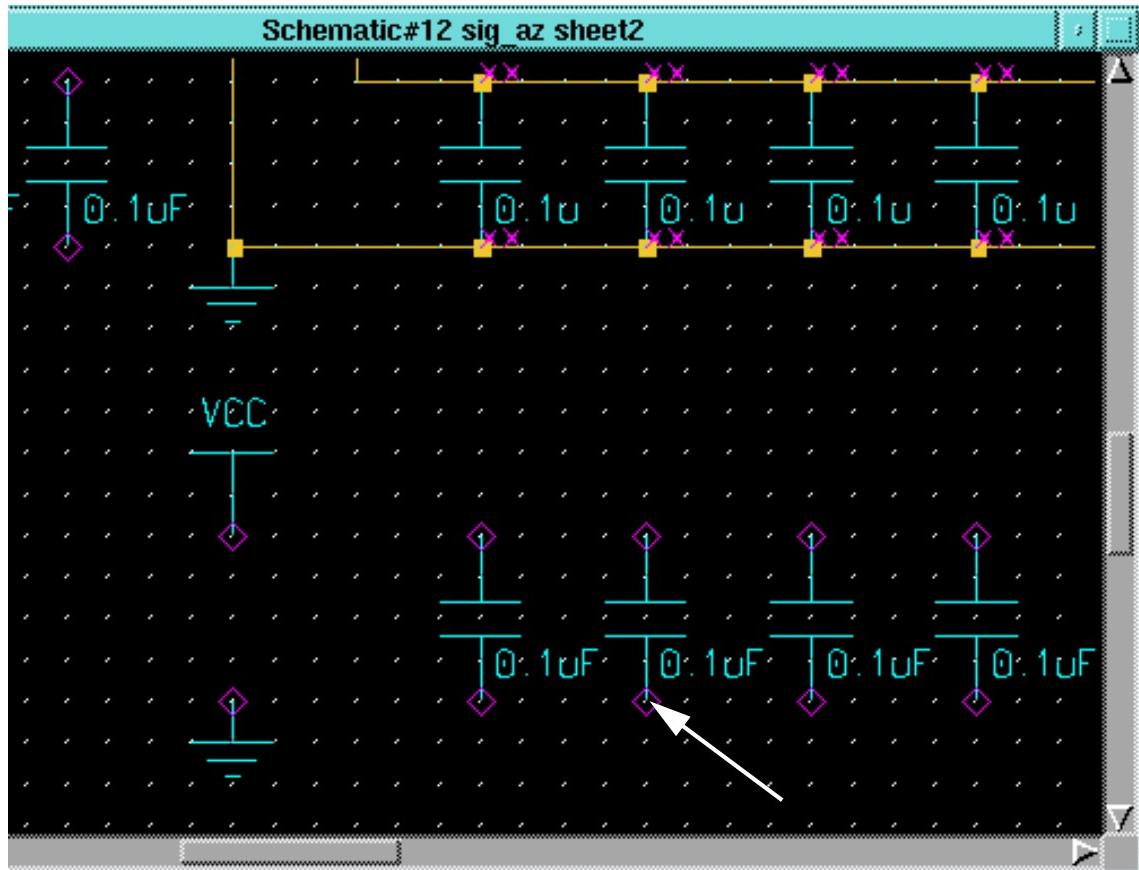
Figure 2-30. Specifying Placement for Copy Multiple

The 15 copies are placed using the direction and inter-copy distance specified when you placed the first copy. When the copies are placed, the prompt bar disappears, and the last copy is selected.

Adding Wires

Next, you add the wire connections to complete the circuitry on this sheet. When adding a wire to a schematic, you add a connection to an existing wire by clicking on the existing wire. If you do not want the new wire to connect to an existing wire, do not click on the existing wire. A new wire only connects to pins or wires that you click on. By default, a connect-dot indicates a connection is made on the schematic. A wire that crosses another net without connecting has no connect-dot.

To make a vertex (corner) in a wire, click on a point that has no other wire or pin. The next point of the wire can then be in any orthogonal direction from the vertex you defined.

If a wire passes over a pin (that you did not click on) a not-dot is placed on the pin. As you saw earlier, you can convert a not-dot to a connection by using one of the **Connections >** items in the **Add** popup menu or the **Edit** pulldown menu.

If you accidentally add a wire that you do not want, you can undo it by first cancelling the Add Wire prompt bar, then choosing **Undo** from a popup menu or from the **Edit** pulldown menu. Or, you can select the wires that are wrong and delete them.

To continue adding wires, click on the **Add_Route Palette > Add Wire** icon again .

The first net connections you add are between the first two capacitors you placed, and from the capacitors to the ground symbol to their right, as shown in Figure 2-31.
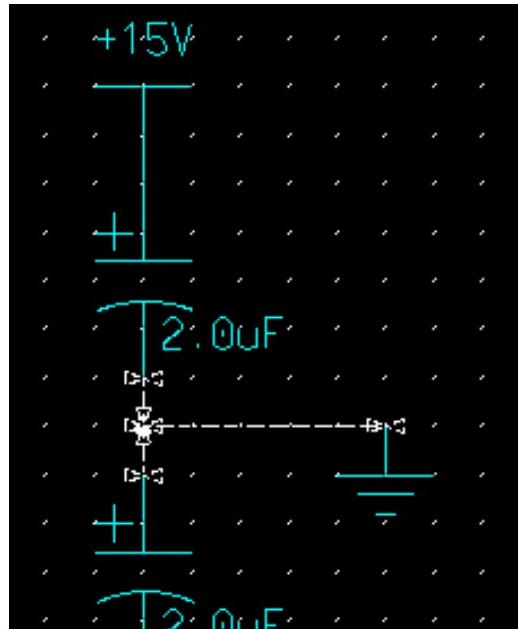

Figure 2-31. Adding Wires
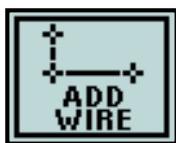
1. Click the Select mouse button on the **Add_Route Palette >
   Add Wire** icon.

   This displays the Add Wire prompt bar.

2. Move the cursor to the bottom pin on the upper capacitor, and click
   the Select mouse button.

3. Move the cursor to the top pin on the lower capacitor, and
   double-click the Select mouse button (or click the Select mouse
   button once, and then **OK** the prompt bar).

   The Add Wire prompt bar is repeated so you can add another wire.

4. Click on the center of the wire just added, then double-click on the
   ground pin, as shown in Figure 2-31.

   Next, you connect the row of three capacitors, then the row of
   capacitors along the bottom of the schematic. You could add one of
   these wires by clicking on each capacitor in the row. An easier and
   faster method for long rows is to click on the first capacitor pin,
   double-click on the last capacitor pin in the row, then connect the
   selected objects. You will use both methods. Figure 2-32 shows
   where to place the first two wires.
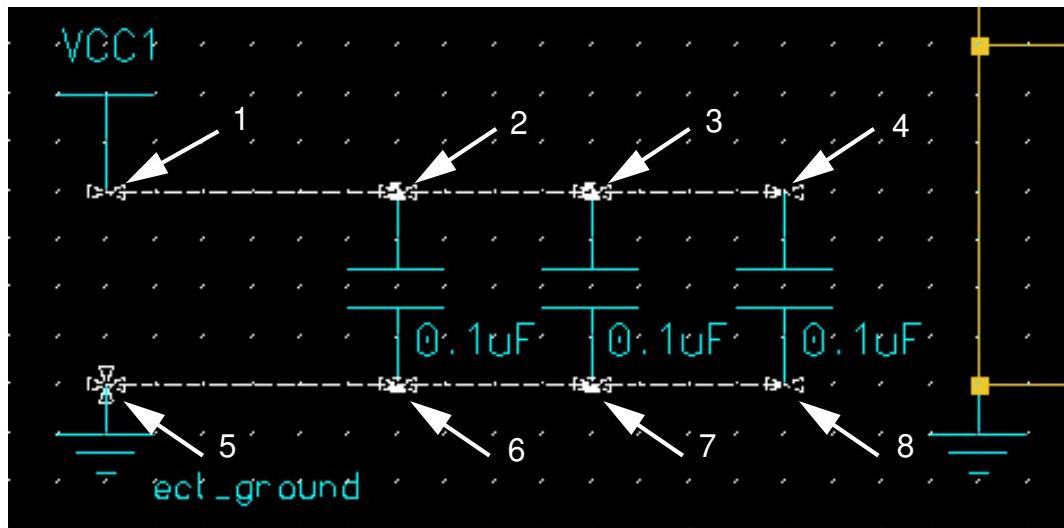
Figure 2-32. Connecting Capacitors: Method 1
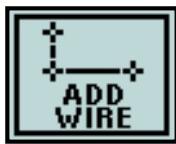
5. If the Add Wire prompt bar is not still displayed, click on the **Add_Route Palette > Add Wire** icon.

6. Using Figure 2-32 as a guide, click the Select mouse button once on each of the first three points, and double-click on the fourth point.

7. Click once on points 5, 6, and 7, and double-click on point 8. Do not cancel the Add Wire prompt bar.

Now you will use the second method to connect the long row of capacitors at the bottom of the sheet. The next steps refer to Figure 2-33.

The figure does not show all of the bottom row of capacitors. They should all be connected together.
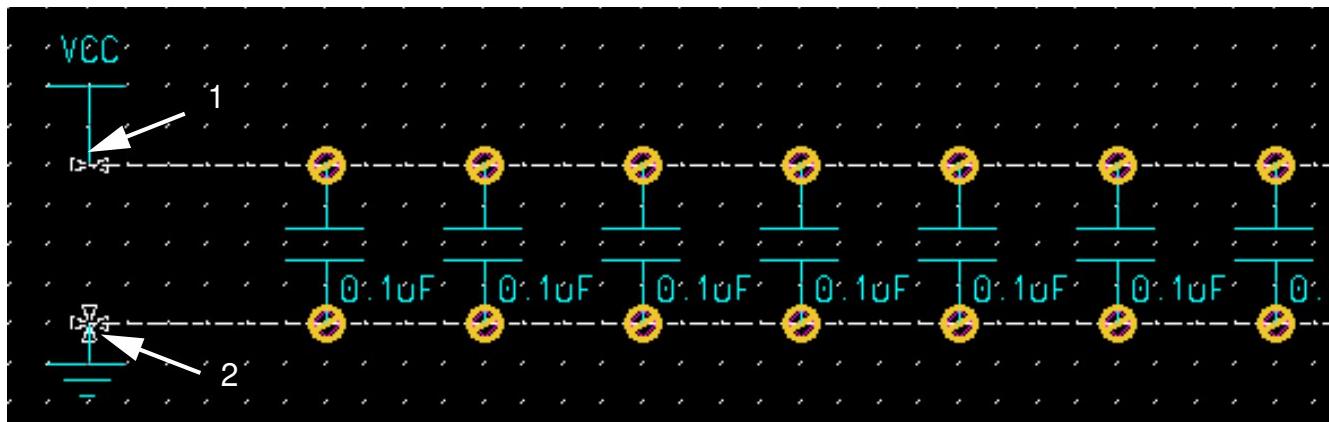

Figure 2-33. Connecting Capacitors: Method 2

8. Press the Unselect All function key.

   You unselect everything now, so that when you connect the selected net segments, nothing unexpected will be connected elsewhere on the sheet.

9. Click the Select mouse button on the **VCC** pin (1), then double-click on the top pin of the right-most capacitor.

   Design Architect places not-dots where you did not click on the capacitor pins to indicate there is no connection.

10. Click the Select mouse button on the **ground** pin (2), then double-click on the bottom pin of the right-most capacitor.

11. **Cancel** the prompt bar.

12. Choose the **[Net] Connections > Connect Selected** popup menu item.

13. Press the Unselect All function key.

    The net segments are connected. After the unselection, the not-dots are replaced by junction dots, as shown in Figure 2-34.
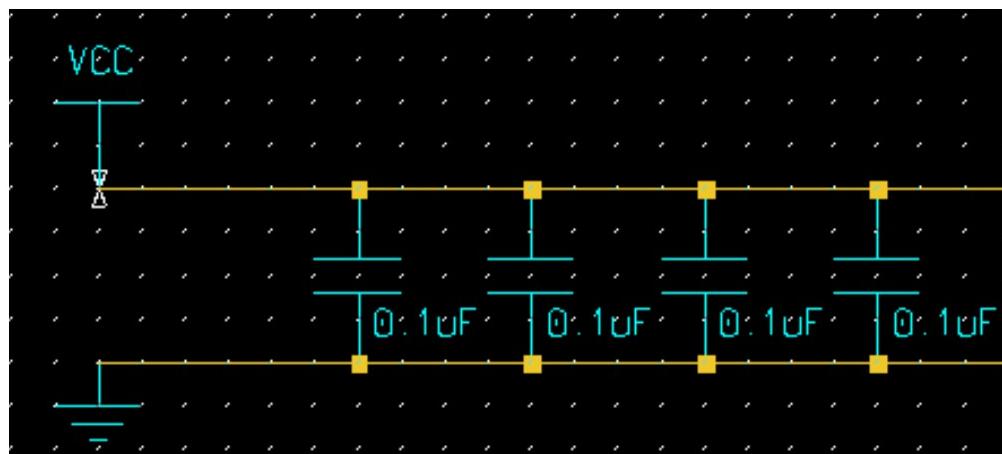

Figure 2-34. Junction Dots

Adding Placement Region Properties

Properties are schematic objects that contain information that can be passed to other tools in the design process. The Placement_region property controls component placement on the circuit board. Components can be placed into named placement region areas during placement in LAYOUT.

The two capacitors you first added to the schematic from the library need to be placed in the analog area of the circuit board. You will now add the Placement_region property to those two capacitors.

1. Select one of the 2.0 *u*F capacitors by clicking on it with the Select mouse button. Next, click on the other capacitor. Check the select count to make sure that only two objects are selected.

2. In the Edit window, choose the **[Instance] Properties > Component Properties > Placement_region** menu item, shown in Figure 2-35.
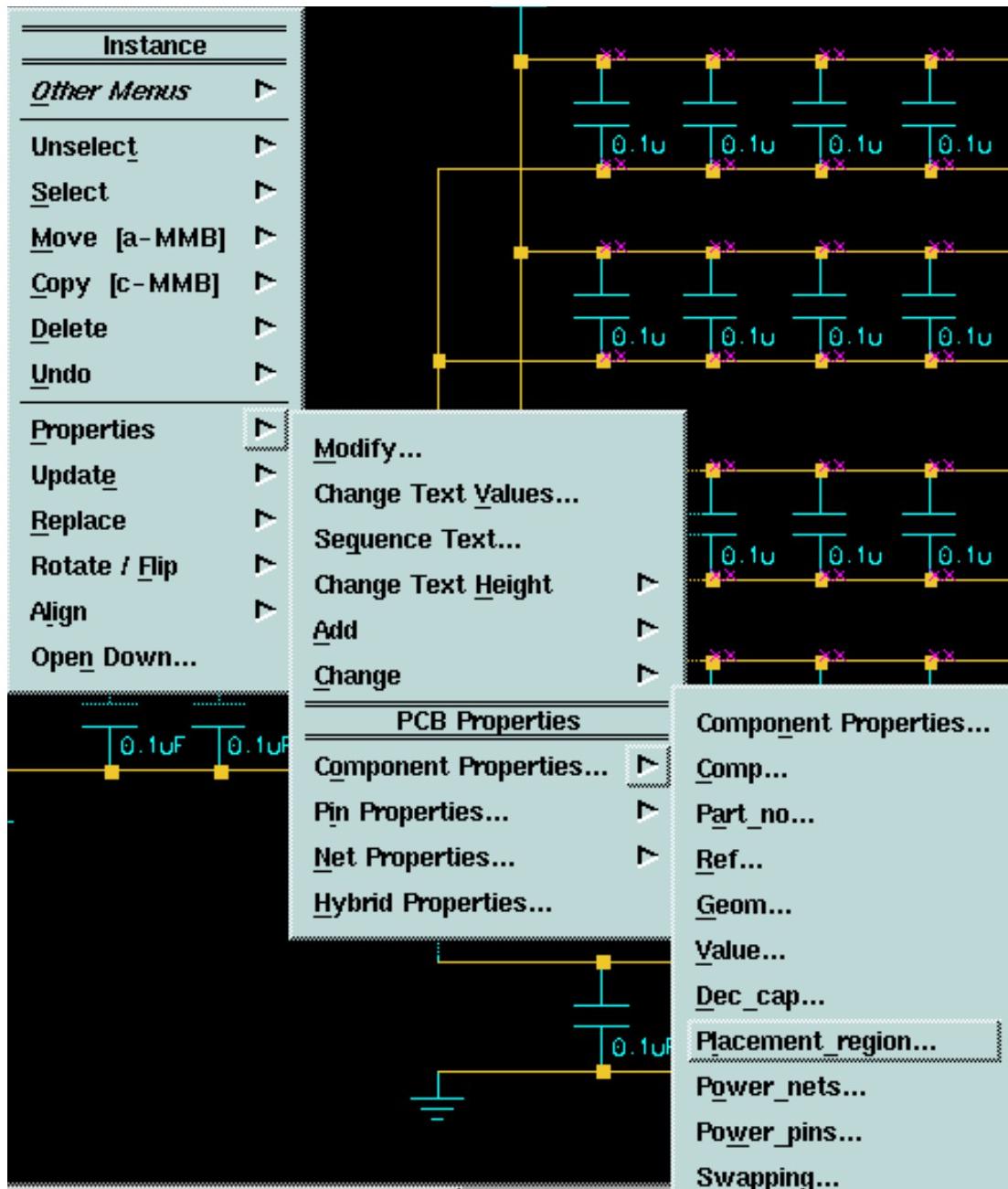
Figure 2-35. [Instance] Properties > PCB Properties Submenu

3. Enter *analog* in the Placement Region Name field in the dialog box, shown in Figure 2-36. If necessary, click on the **Visibility** button to unhighlight it to indicate that this property should not be visible on the schematic, then **OK** the dialog box.
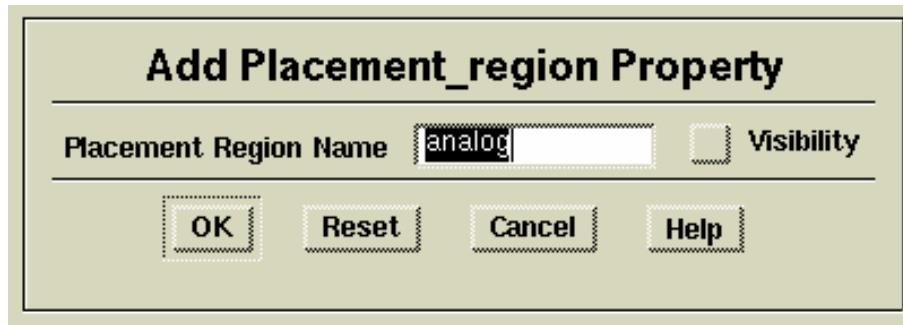
Figure 2-36. Add Placement_region Property Dialog Box

    4. With the capacitors still selected, choose the **Report > Object >
    Selected** menu item.

    A Report window is displayed, listing the characteristics of selected
    objects, including properties and property values. In the list of
    properties, you should see Placement_region with a value of
   analog, as shown in Figure 2-37.

Figure 2-37. Report Window Showing Property Values

    5. Close the Report window, and unselect all objects (use the Unselect
    All function key).

Checking and Saving the Schematic

The final step before saving a schematic sheet is to perform a data check on the sheet. The checking function checks for interconnected wires, overlapping symbols, and other problems. Because the circuitry on this sheet is within a frame, you will set a value for the frame expression variable (layout) so the sheet can be evaluated.

1. Choose the **Check > Parameters > Set** pulldown menu item shown in Figure 2-38.
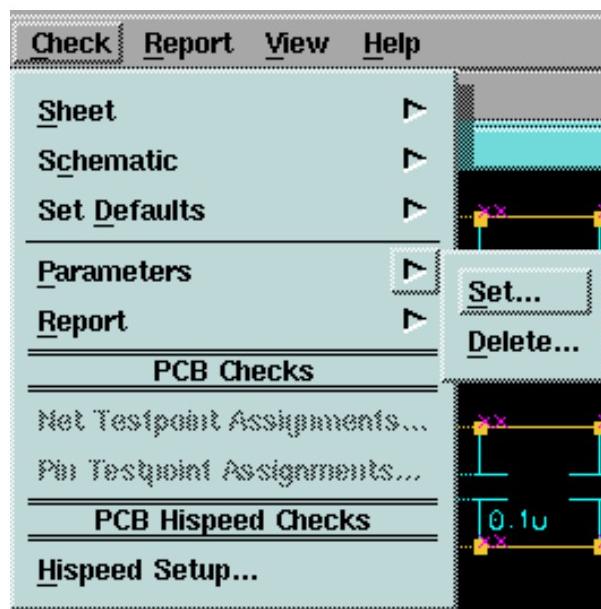
Figure 2-38. Check > Parameters Menu

2. Complete the Set Parameters dialog box, as shown in Figure 2-39.
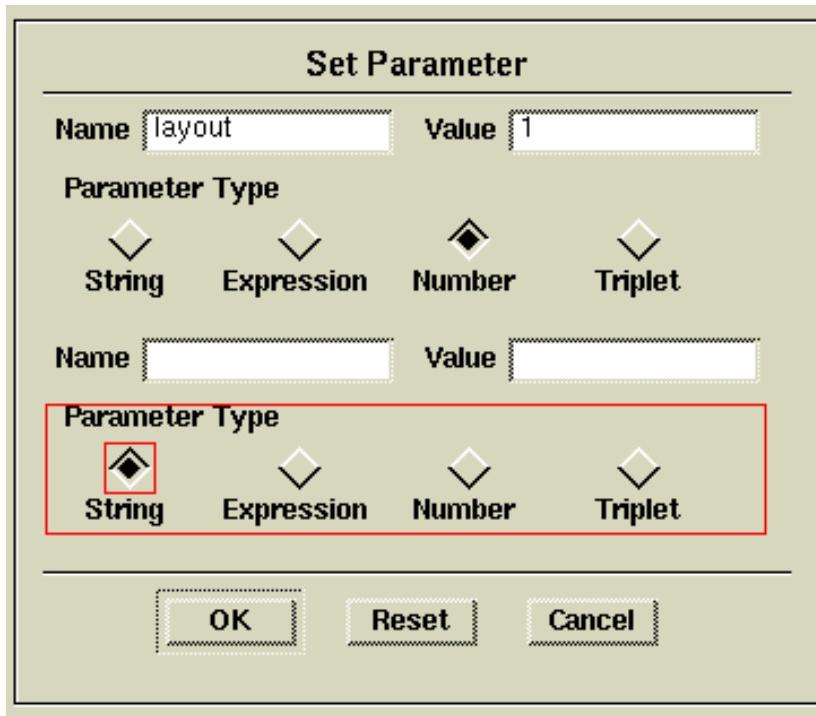
Figure 2-39. Set Parameter Dialog Box

3. Choose the **Check > Sheet** menu item.

A Notepad Report window appears with the results of the checking. You might find warnings about unconnected pins and/or dangling vertices. These warnings are not a problem for this design. Schematic sheet warnings are allowed by Mentor Graphics applications; errors are not allowed. If you find errors in the listing, check with your instructor or system administrator for specific solutions.

If there are warnings or errors, the object that caused each error or warning will be identified by its handle (such as I$256 for an instance, N$235 for a net, and so on). To help you locate the problem area of the schematic, you can select the object causing the problem by using the Select mouse button to click on the handle (such as I$256) in the Report window. The corresponding object will be selected in the schematic so you can see it. Be sure to check the design again after correcting any errors.

An example of a check report is shown in Figure 2-40.

```
Check#5  sig_az:Sheet

Check Sheet "sig_az/schematic/sheet2"
Check SymbolPins --------   0 errors   0 warnings   (MGC-required)
Check Overlap -----------   0 errors   0 warnings
Check NotDots -----------   0 errors   0 warnings
Check Closedots ---------   0 errors   0 warnings
Check Dangle ------------   0 errors   0 warnings
Check INIT Properties ---   0 errors   0 warnings
Check Owner -------------   0 errors   0 warnings
Check Instance ----------   0 errors   0 warnings   (MGC-required)
Check Special -----------   0 errors   0 warnings   (MGC-required)
Check Net ---------------   0 errors   0 warnings   (MGC-required)
Check Frame -------------   0 errors   0 warnings   (MGC-required)

 "sig_az/schematic/sheet2" passed check :   0 Errors, 0 Warnings
```

Figure 2-40. Check Report

4. After you correct any errors, close the Report window by choosing the **Window > Close** menu item shown in Figure 2-41.

```
Restore     "sig_az,
            lPins ---
Move        ap ------
Size        ts ------
            dots ----
Minimize    e -------
Maximize    Propertie
            -------
Pop         nce -----
            al ------
Close       --------
            --------
```
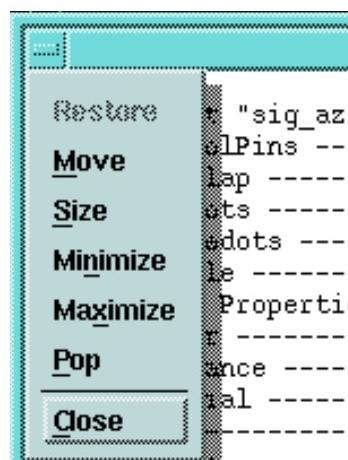
Figure 2-41. Window > Close Menu

5. Choose the **File > Save Sheet** menu item shown in Figure 2-42.
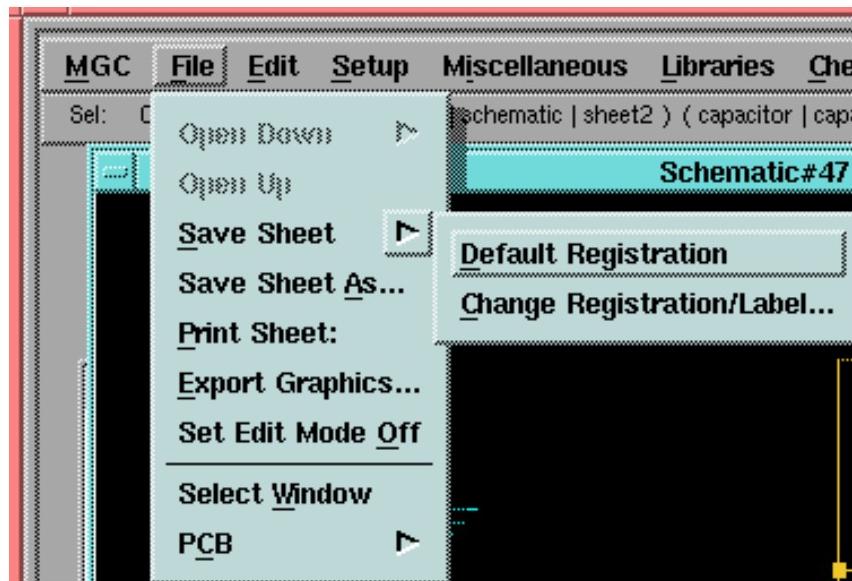

Figure 2-42. File > Save Sheet Menu

6. Close the schematic sheet by choosing **Window > Close** in the Schematic Editor window, but do not close the Design Architect Session window.

Planning for Split Power Planes

Some of the components in this design require different voltage levels. In Module 3: "Creating PCB Design Geometries", when you define the layers of your board, you will create split power planes instead of creating a power layer for each voltage. A split power plane is a single layer of a board with separate power fill areas connected to different power nets. If the components with the same power connections are grouped together, it is easier to create the power fill areas in FabLink.

In this part of the lab, you will use the Placement_region property to group components that need 15 volts.

1. In Design Architect, click on the **Palette > Open Sheet** icon.

2. In the Open Sheet dialog box, navigate to the lab data:

   your_path/**training/board_new/mod2**

3. Click on **sig_az**, and **OK** the navigator. Leave **sheet1** as the sheet name, and **OK** the dialog box.

   Sheet1 of the design is displayed in a Schematic window.

4. Click the **Maximize** button in the upper right-hand corner of the Schematic window.

5. Select the component labeled **DATA_IO** by clicking the Select mouse button on it (see the arrow in Figure 2-43).
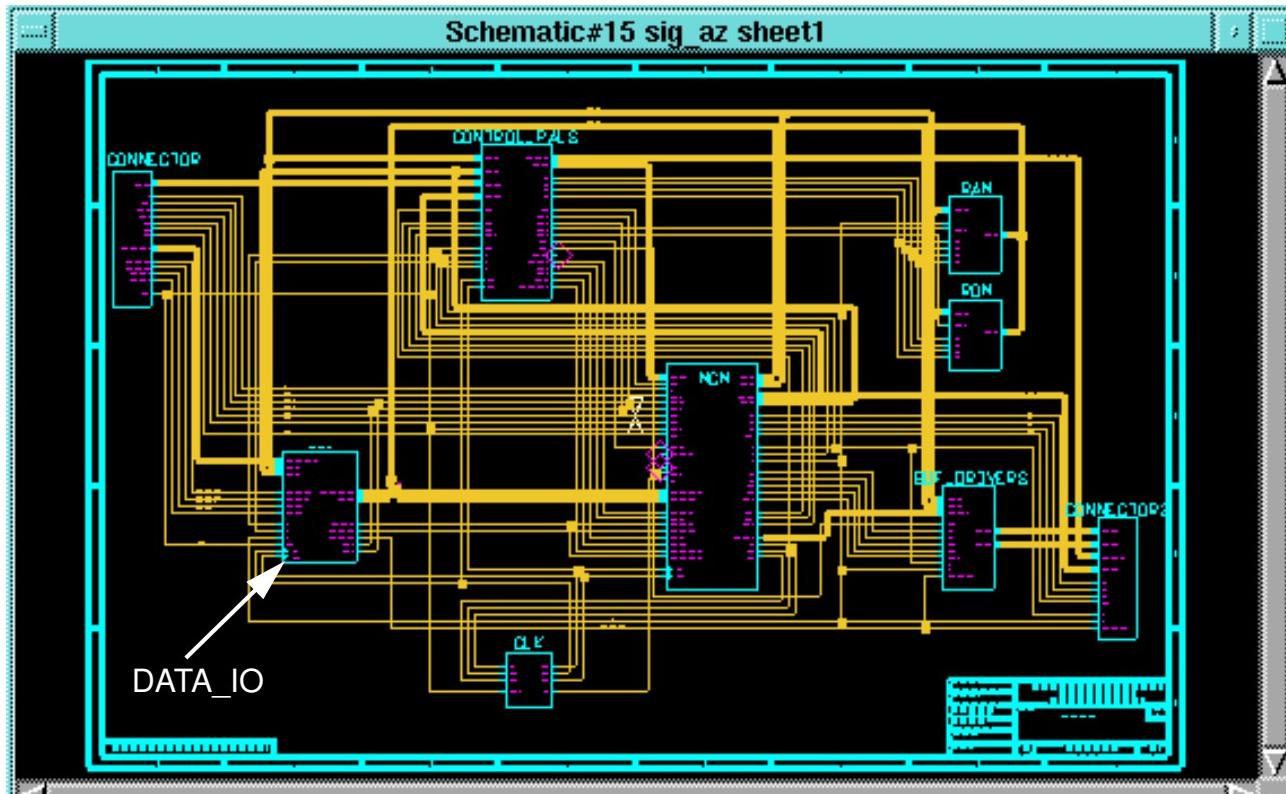
Figure 2-43. DATA_IO Component on Sheet1

6. Choose the **File > Open Down > Choose Model** menu item shown in Figure 2-44.

Figure 2-44. File > Open Down Menu

In the Open Down dialog box, click on **schematic** in the list of models, and **OK** the dialog box.

The DATA_IO sheet is displayed in a new window.

7. Maximize the newly opened window and select the component labeled a_d_block.

Figure 2-45 shows the DATA_IO sheet with the a_d_block selected.
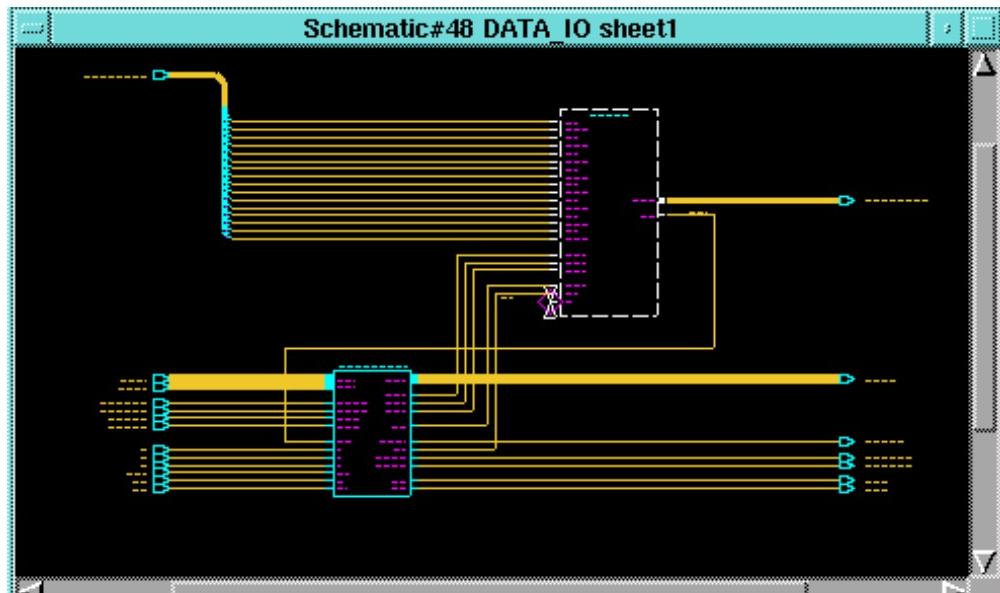
Figure 2-45. DATA_IO Sheet

8. Again, choose the **File > Open Down > Choose Model** menu item, click on **schematic** in the list of models, and **OK** the dialog box.

   The a_d_block sheet is displayed in a new window.

9. Maximize the window and click the Select mouse button on each of the four main components on the a_d_block sheet, identified by arrows in Figure 2-46.
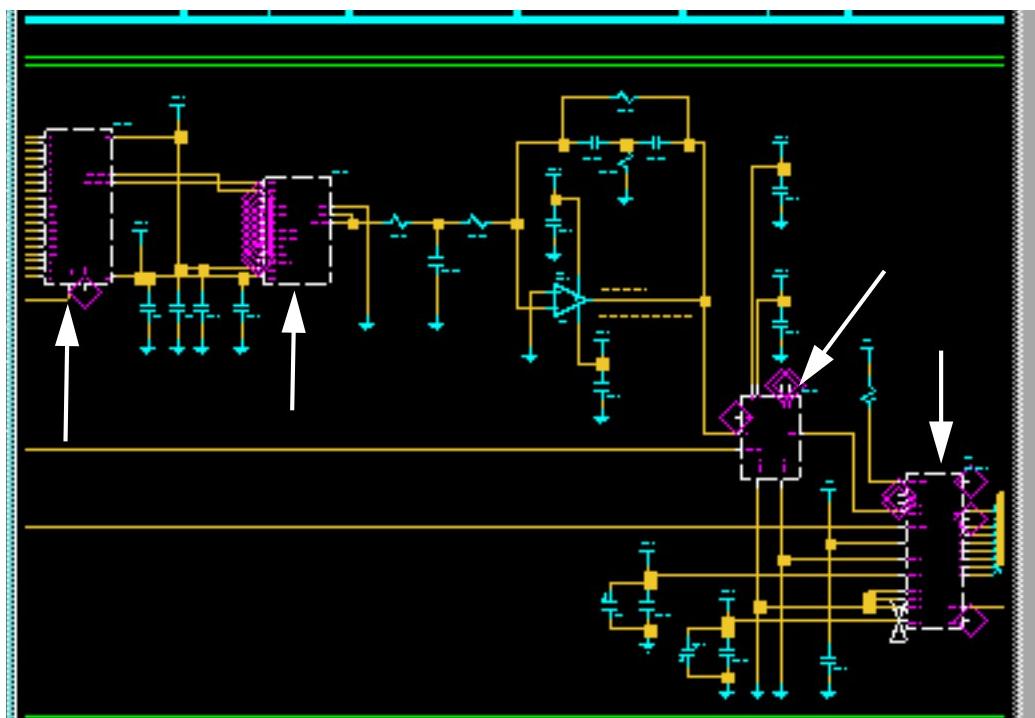

Figure 2-46. Selected Components on A_D_BLOCK

10. Choose the **[Instance] Properties > Component Properties > Placement_region** popup menu item.

11. Enter **V15** in the dialog box and be sure **Visibility** is turned off, as shown in Figure 2-47. **OK** the dialog box.

Figure 2-47. Add Placement_region Property Dialog Box

The four components just assigned the Placement_region property will be physically grouped together on the board in an area defined as *V15*.

Selecting Objects by Property

Sometimes it is helpful to be able to select objects that have a particular property. The following procedure describes how to select by property when you do not know the property value.

1. Press the Unselect All function key to be sure nothing on the sheet is selected. Choose the **[Add] Select > By Property > Name-Value-Type** popup menu item shown in Figure 2-48.
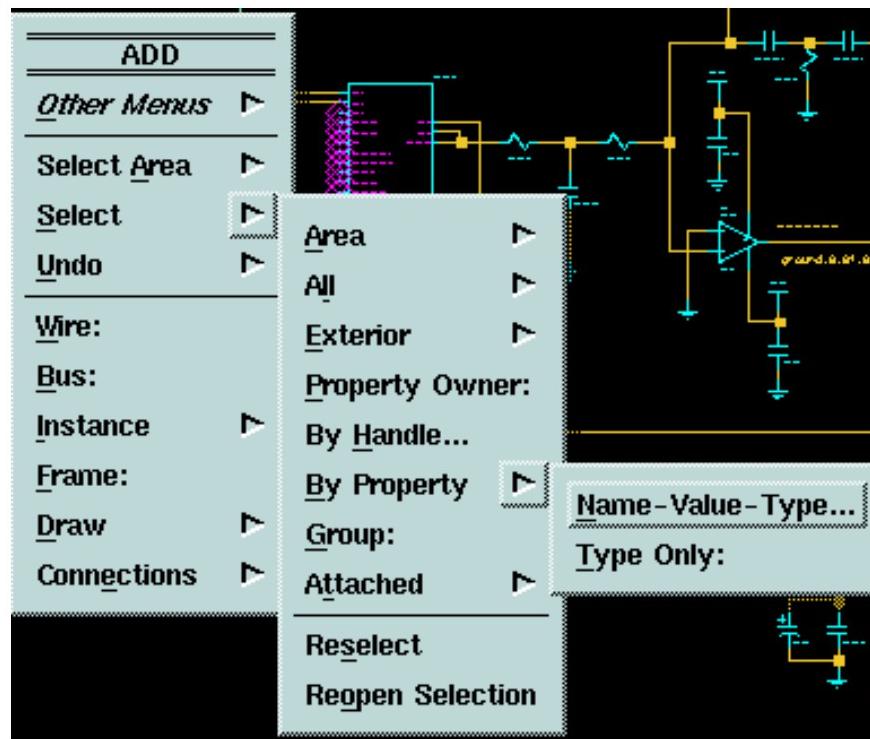
Figure 2-48. Select > By Property Menu

This displays the Select By Property dialog box.

2. Enter **placement_region** for the property name and choose **Union** from the selection mode choices. **OK** the dialog box.

In the Select By Property dialog box, *Union* is the default property selection mode. If more than one property is specified in the dialog box, all design objects having any of the specified properties are selected. *Intersection* selects only the objects having all specified properties. *Text* selects the property text, rather than the property owner.

All objects that have the Placement_region property assigned to them are selected. If no objects have the specified property, a message is displayed in the message area saying nothing is selected.

3. Choose the **Report > Object > Selected > All** menu item.

4. Scan the displayed report to find the property value. Figure 2-49 shows an example of property values in a report.

Figure 2-49. Report Showing Property Values

5. Close the Report window.

6. Unselect all objects by pressing the Unselect All function key.

Using Object Handles

There are many system functions you can use to find information. System functions are generally not accessible through a menu. You invoke them by typing the function in the Edit window. When you begin typing, a popup command line appears to display what you are typing.

Design Architect system functions are documented in the *Design Architect Reference Manual*. Also see the *Common User Interface Reference Manual* for functions that are available in all applications.

Many system functions require an object handle as an argument. You can get an object handle from a report, or you can use a system function to get it.

Use the following steps to get an object handle and find the value of the Placement_region property on a component.

1. Select any one component on the a_d_block sheet. The a_d_block sheet is shown in Figure 2-46 on page 2-55.

2. Type the following in the popup command line in the Edit window:

> **Schematic#11** $get_select_handles()

The popup command line appears when you begin typing. The object handle appears in the transcript.

3. Choose the **MGC > Transcript > Show Transcript** pulldown menu item shown in Figure 2-50.


Figure 2-50. MGC > Transcript Menu

The transcript lists an instance handle, such as:

// ["I$1"]

Now that you have the object handle, you can use system functions to obtain information.

4. Type the following in the Edit window to get the Placement_region property value on the selected component:

$get_object_property_attributes("I$1", "placement_region", @value)

5. The transcript shows the value of the specified property:
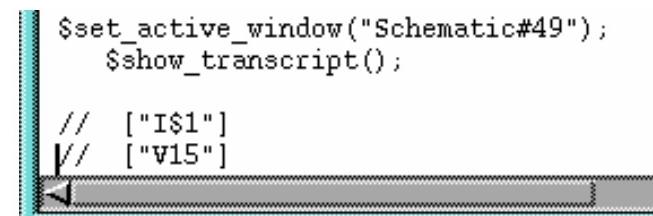
```
$set_active_window("Schematic#49");
    $show_transcript();

//  ["I$1"]
//  ["V15"]
```

// ["v15"]

If the property had no value, the transcript would show:

// [" "]

6. Close the Transcript window.

Saving and Closing

You need to save and close the schematic sheets. Because you made changes to the a_d_block sheet, it needs to be checked before closing.

1. Activate the a_d_block sheet, and choose the **Check > Sheet** pulldown menu item.

   There should be no errors.

2. Close the Report window using the **Window > Close** menu item.

3. Choose the **File > Save Sheet** menu item. After saving the sheet, close the Schematic window.

4. Check and save the a_d_block schematic. Close the window.

5. Check and save the DATA_IO and sig_az schematics. Close the windows.

6. Close the Design Architect Session window.

Congratulations! You have completed the "Editing a Schematic for PCB" lab exercise. Continue with Lesson 3 "Creating Design Viewpoints" in this workbook.

Lesson 3
Creating Design Viewpoints

Preparing a design for PCB includes creating a PCB viewpoint, creating an engineering viewpoint, and creating and connecting back annotation objects to the viewpoints in which to place design changes.
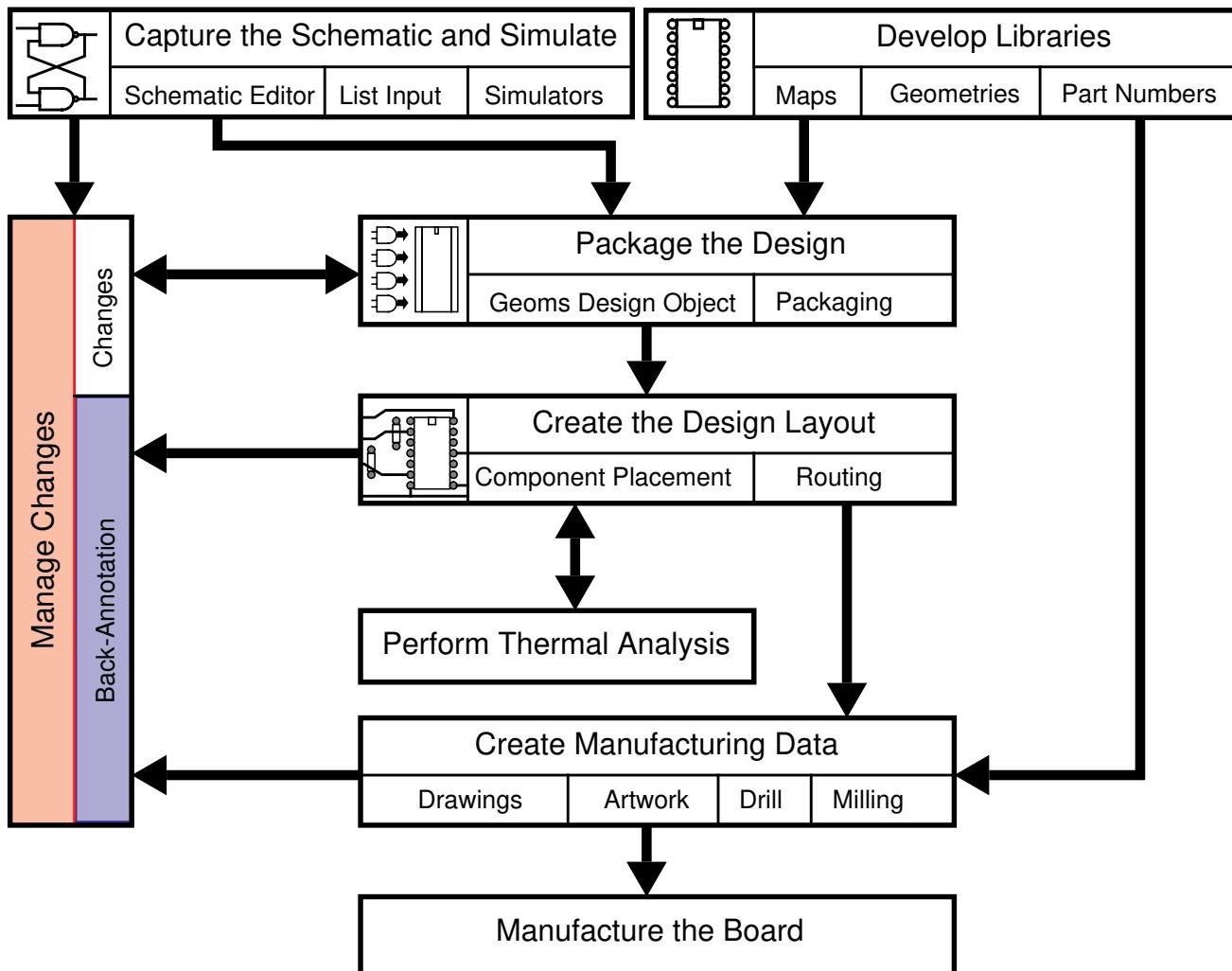

Figure 3-1. Board Process Flow Chart

Objectives

This module discusses viewpoints and back annotation objects, what they are, why they are important, and how to create them in the Design Viewpoint Editor (DVE). You will learn about primitive properties, default visible properties, and how to define and add a visible property. Finally, you will learn how to check the design in DVE.

After completing this module, you should be able to do the following:

- Explain the purpose of a design viewpoint.

- Describe the categories of design rules in a design viewpoint.

- Create viewpoints using DVE.

- Create and connect back annotation objects.

- Check a design in DVE.

Process for Viewpoints and Back Annotation Objects

You will use the following process to create two design viewpoints, and create and attach two back annotation objects in DVE. Each step is covered in detail in this lesson, and you will follow this process in the lab exercise.

1. Create and save a PCB design viewpoint.

2. Create and save an engineering design viewpoint.

3. Connect back annotation objects to the engineering viewpoint in the proper order.

You do not need to use the Move menu item to position components this time, as you are already prompted for a location.

What Is a Viewpoint?

A *design viewpoint* is a set of design rules by which an application evaluates that design. The design rules determine the values of variables in the design, and the level of hierarchy at which components are evaluated. A design viewpoint is not a copy of a design; it is the device by which correct data, in the form of an evaluated design, is presented to downstream applications.

Viewpoints containing setup information can be created by applications that need an evaluated design as input. For example, if a design viewpoint does not exist, one is automatically created during invocation of a downstream application such as QuickSim II or PACKAGE.

You can have multiple viewpoints for most downstream tools. However, you can have only one PCB viewpoint, because PCB tools write viewpoint and back annotation information into the PCB design database.   PCB tools use the default viewpoint name *pcb_design_vpt*.

In Design Architect, when you open a sheet in the context of a design (using Open Design Sheet), you are viewing the design through a viewpoint. If you open a design sheet in Design Architect with the viewpoint name *pcb_design_vpt* and that viewpoint does not exist, Design Architect creates a design viewpoint containing the default setup information for PACKAGE.

You can create a custom design viewpoint using the Design Viewpoint Editor. The Open Design Viewpoint dialog box is shown in Figure 3-2.
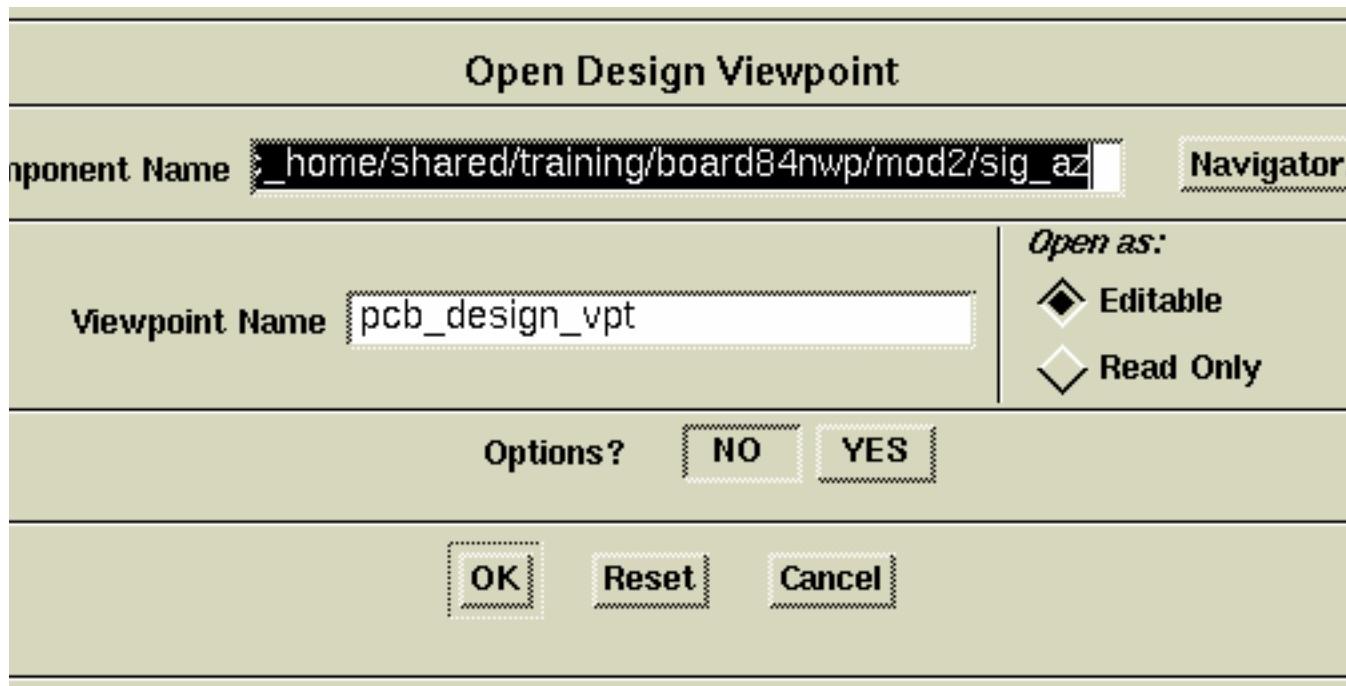
Figure 3-2. Open Design Viewpoint Dialog Box

If you use default settings in the Design Viewpoint Editor, the PCB
design viewpoint it creates behaves no differently than a PCB design
viewpoint created by PACKAGE using default settings.

The viewpoint has three sets of rules: primitives, parameters, and
visible properties. The design rules in a viewpoint are discussed next.

Primitives

Design evaluation begins at the highest level of hierarchy, and progresses down through the design until stopping points, called *primitives*, are reached. The depth of design evaluation depends on which downstream application you plan to use.

The default PCB design viewpoint defines all symbol instances having a Comp property with any value as primitive because this defines the component, or physical, level of the object. All component data that PCB layout applications require from the schematic are placed as properties at this level.

Simulators use the Model property to define primitive components. By adding Model property values to the primitive list, you can simulate portions of your design. Another method is to set the Model property value to *null* for components that you want to exclude from evaluation.

You set which instances are primitive by adding a primitive rule to the design configuration, specifying a property name and, optionally, a value. If no primitive rules are specified, evaluation stops at components that have no functional models.

You can also exclude certain instances from being considered primitive. For example, suppose you are creating a PCB viewpoint. You have three functional blocks in your design, and you have assigned the Comp property to each functional block with the values *bl_1*, *bl_2*, and *bl_3*, respectively.

To evaluate only bl_1, you would click on the **Palette > Add Prim** icon and complete the Add Primitive dialog box as shown in Figure 3-3.
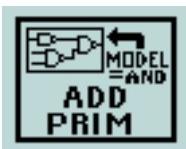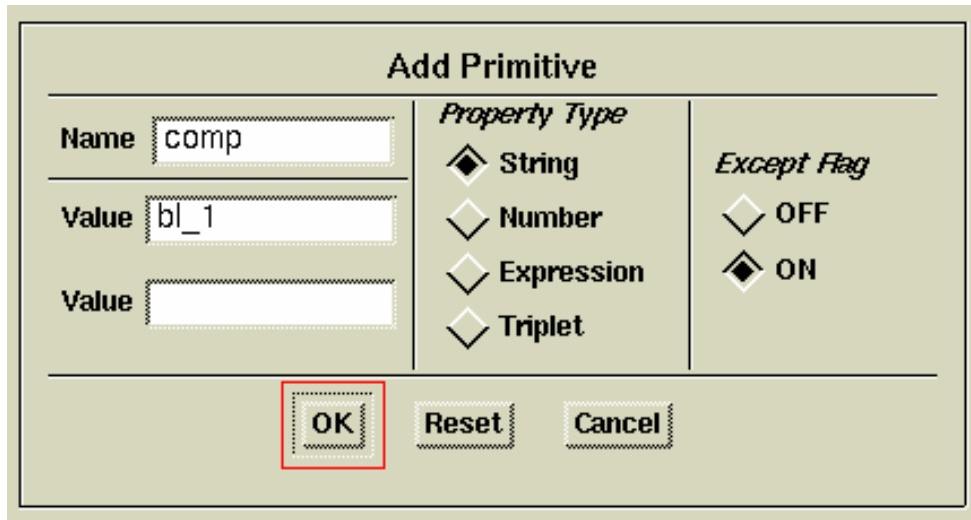
Figure 3-3. Add Primitive Dialog Box

If a schematic view window is open, it is redrawn, and the Design Configuration window is updated to show the new primitive rule.

To change the value of a primitive, select the primitive in the Design Configuration window. Choose **Design Configuration > Edit** and complete the dialog boxes.

To delete a primitive, select it in the Design Configuration window and choose **Design Configuration > Delete**.

Parameters

A *parameter* is a value for a variable in a property value that is resolved outside of the component. The value of the variable may be supplied in Design Architect, or through a technology file, or by adding a parameter in the design viewpoint.

During design evaluation, if an unresolved property variable is found, the design tree is searched for the value. If no value is found in the design tree, the design viewpoint is searched for a value. Therefore, you must add parameters prior to design evaluation.

Frame expressions use property variables. The following are examples of frame expressions:

- FOR I := 1 TO N

  The circuitry within the frame having this expression is repeated N times. N must have a value so the design can be evaluated.

- IF LAYOUT == 1

  This frame expression is used in your lab design. For PCB tools, the circuitry within the frame is used; for other tools, it is not used.

You can also use CASE frames in schematics to conditionally use portions of the design.
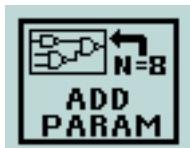
To add a parameter to the design viewpoint, click on the **Palette >Add Param** icon, and complete the dialog box shown in Figure 3-4.
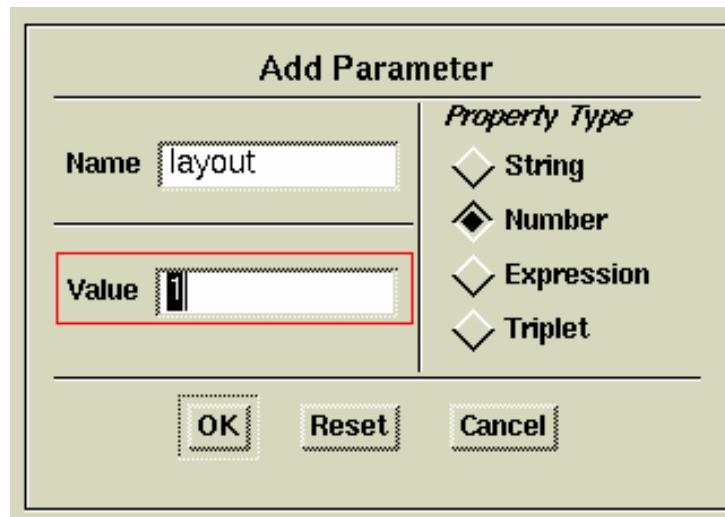
Figure 3-4. Add Parameter Dialog Box

To change a parameter, select it in the Design Configuration window. Choose **Design Configuration > Edit** and complete the dialog boxes.

To delete a parameter, select it in the Design Configuration window and choose **Design Configuration > Delete**.

Visible Properties

The *visible property* rule specifies which properties are recognizable by downstream tools, such as PACKAGE. When you create a viewpoint, you choose one of the applications from the **Setup** menu, shown in Figure 3-5.
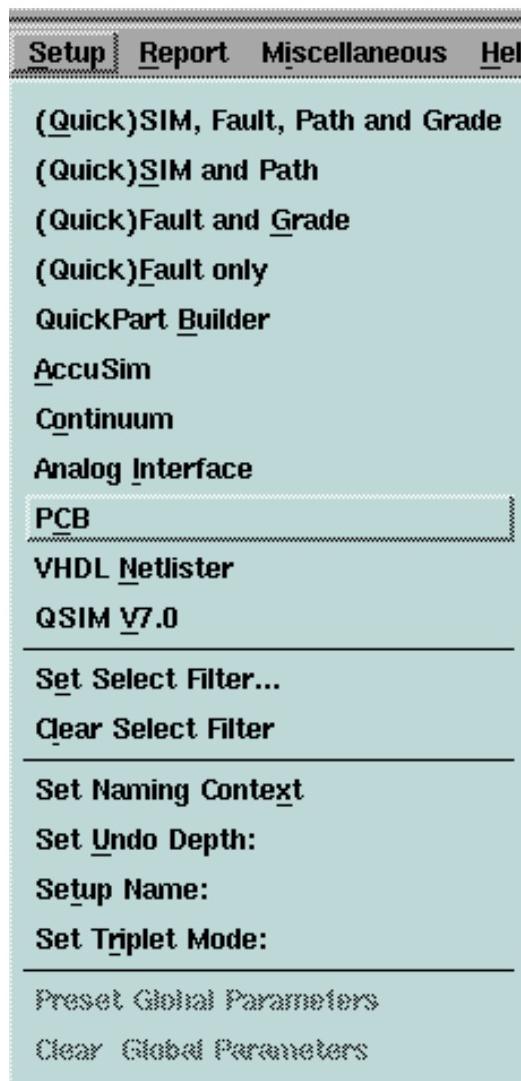

Figure 3-5. DVE Setup Menu

DVE creates a default list of properties for the viewpoint, depending on which application you chose. For example, a PCB viewpoint needs properties such as Placement_region and Ref, while a QuickSim

viewpoint needs other properties such as Rise and Fall. The purpose of each tool is different, so they each look at different aspects of a design. You should add all user-defined properties to your PCB viewpoint, and also to the *pkgconf* design object.

For information about *pkgconf*, refer to "Data Preparation Procedures" in the *PCB PACKAGE User's Manual*.

To define visible properties, choose the **Design Configuration > Add > Visible Property** popup menu item. In the dialog box, shown in Figure 3-6, click on the owner type and enter as many property names as desired. Typically, you will issue this rule three times, once for each owner type (Instance, Net, Pin) with a list of property names for that owner.
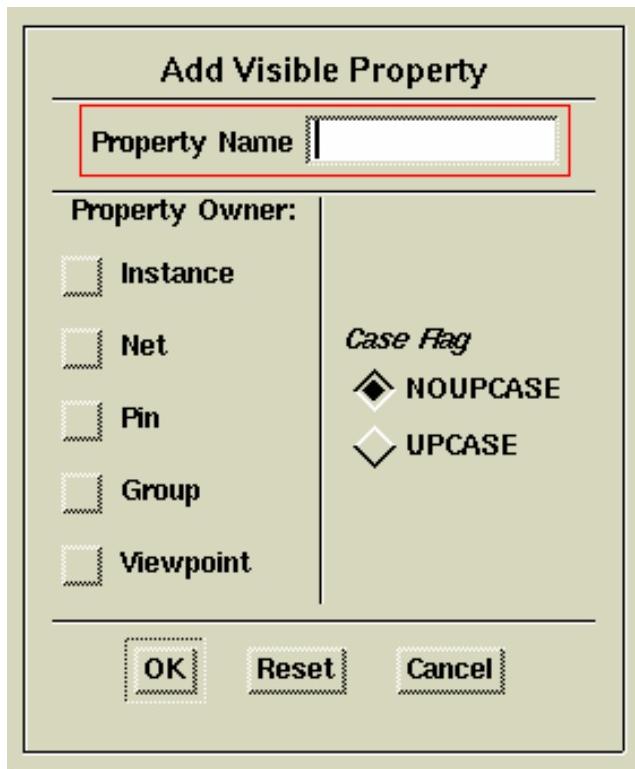

Figure 3-6. Add Visible Property Dialog Box

Properties attached to pins migrate to nets when the design is seen through a design viewpoint. Therefore, if you want these pin properties visible to nets, you must include the pin property names in the list of properties owned by nets.

To edit an existing visible property rule, select that rule in the Design Configuration window, choose **Design Configuration > Edit**, and complete the dialog box information.

To delete a visible property rule, select the rule and choose **Design Configuration > Delete**. The specified visible property is deleted from the design viewpoint and will not be recognized by tools using that viewpoint.

Substitutes

Substitution uses the value of one property for the value of another property. Substitutes temporarily change a property value for complete evaluation of the design. As a general rule, you don't need substitutes; use variables or expressions in your property values instead.

Annotations

Annotations are property name/value pairs that pass design information from one application to another during the design cycle. Annotations may be passed both forward (forward annotation) and backward (back annotation).

Forward Annotation

Forward annotation is the process of passing property information forward from Design Architect to the simulation and PCB applications. As you create and edit your schematic in Design Architect, you add properties to nets, instances, and pins that influence how another application will process those objects. The new or modified values of those properties are forward annotations; that is, the values are brought forward from the schematic to the PCB tools.

For example, if you have four gates that are packaged two to a component, you can assign the Ref property to each of the four gates to specify which gates are packaged together. In Design Architect, you select two symbol instances (representing the gates) that you want packaged together and add the Ref property with the same value, such

as U23. Then you select the other two instances and add the Ref
property with another value, such as U24. The Build program in
PACKAGE places the first two instances in one component, and the
second two instances in another component, and assigns U23 and U24,
respectively, as the reference designators.

Back Annotation

Back annotation is the process of transferring packaging and layout
information from PCB tools back to the schematic. The information
you transfer consists of either properties that were originally on the
schematic and changed or deleted in a PCB tool, or properties that you
added in a PCB tool.

When you back annotate a design, the additions and modifications are
stored in a back annotation object (file) which is connected to the
design viewpoint. You should back annotate the PCB viewpoint at
every major stopping point in your design cycle. For example, back
annotate before you exit each PCB application. Be sure the properties
you back annotate are listed among the visible properties in the PCB
design viewpoint.

To back annotate a design from PACKAGE, LAYOUT, or FabLink,
choose the **File > Back Annotate** menu item.

When you back annotate, the modified property values are shown on
each gate as you view or edit the schematic in the context of your
design. To access a schematic in the context of a design, choose the
[Session] Open Design Sheet popup menu item, or click the Select
mouse button on the **Palette > Design Sheet** icon in Design Architect.

For more information about back annotation, refer to section "Back
Annotating" in the *PCB Products Design Reference Manual*.

For information about design viewpoints, refer to section "Overview
of a Design Viewpoint" in *A Guide to Design Process and Database
Concepts*, and refer to the *Design Viewpoint Editor (DVE) User's and
Reference Manual*.

Latching the Design

When both schematic design and board layout are happening in parallel, the layout designer needs to have a stable version of the design and be isolated from changes made by the electrical engineer. To keep a stable version, you *latch* the PCB design viewpoint in DVE.

Latching a viewpoint locks the versions of specified objects, or of all objects referenced by the viewpoint, such as symbols, sheets, and models, so that the same versions of all objects are referenced every time the viewpoint is used. Latched versions of objects cannot be deleted by any tool.

The layout designer can update the latch at any time. Updating the viewpoint's latch means the currently latched versions of objects are unlatched and the most recent versions of those objects are latched. Updating the latch enables the layout designer to see the edits made by the electrical engineer.

For information about latching, refer to the *Design Viewpoint Editor User's and Reference Manual* and to *Board Station for New Users Training Series, Module 8: Managing Design Changes.*

Creating a PCB Design Viewpoint

Because the PCB applications and the QuickSim simulation applications require different levels of primitiveness and different visible properties, you need two design viewpoints. You create a PCB design viewpoint first, and then an engineering design viewpoint.
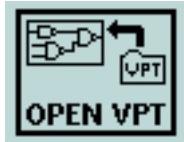
You can invoke DVE from the Design Manager by clicking the Select mouse button on the DVE icon in the Tools window. When the DVE Session window displays, click the Select mouse button on the **Palette > Open Vpt** icon.

In the Open Design Viewpoint dialog box, enter your design component pathname, or select the design using the Navigator, and **OK** the dialog box.

You can enter a viewpoint name in the dialog box, if desired. If not specified, *default* changes to *pcb_design_vpt* when you save the viewpoint. *Pcb_design_vpt* is the default viewpoint name that PCB applications use.

To create a default PCB viewpoint with visible properties added and the Comp property defined as primitive, choose the **Setup > PCB** pulldown menu item. Alternatively, you can click on the **Palette > Setup Vpt** icon, which displays the dialog box shown in Figure 3-7. DVE also creates a back annotation object and connects it to the viewpoint.
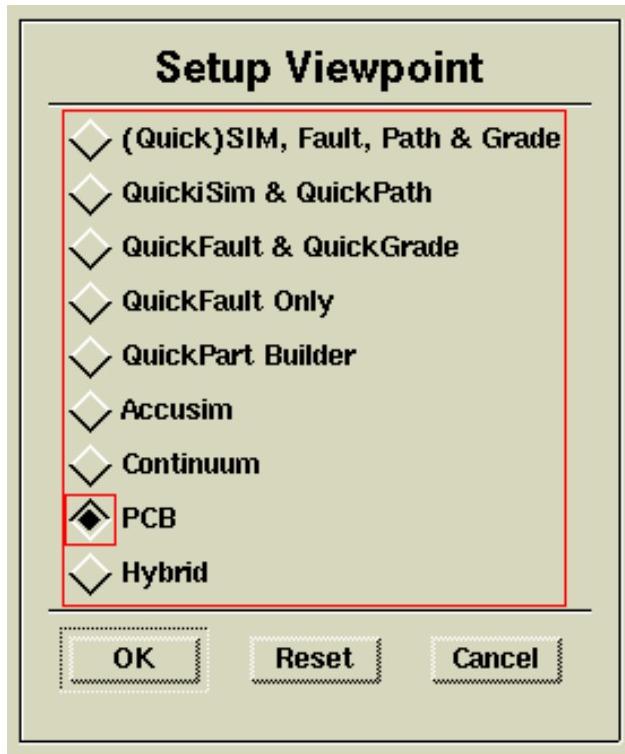
Figure 3-7. Setup Viewpoint Dialog Box.

Now you can customize the design viewpoint to fit your particular design. You need to specify in the viewpoint any properties you have added that you want visible to PACKAGE or other PCB tools. You also need to add any pin properties to the list of net-owned visible properties if you want the property values propagated to the nets. Use the **Design Configuration > Add > Visible Property** popup menu item to accomplish these tasks.
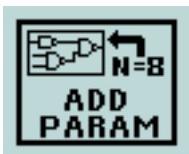
To define values for any variables that would otherwise be unresolved when an application evaluates the design, click on the **Palette > Add Param** icon, and enter values in the dialog box that appears.

Perform the following steps to change the primitive level for design evaluation:

1. Click on the **Palette > Add Prim** icon.

   This displays the Add Primitive dialog box.

2. Enter the property name for the new primitive level.

3. Optionally, enter a property value, and **OK** the dialog box.

After editing the design viewpoint, save your results by clicking on the **Palette > Save Design** icon. Close the viewpoint by clicking on the **Palette > Close Vpt** icon.

Creating an Engineering Design Viewpoint

The process for creating a engineering design viewpoint is similar to that for a PCB viewpoint. After closing the PCB design viewpoint in DVE, click the Select mouse button on the **Palette > Open Vpt** icon.

In the Open Design Viewpoint dialog box, enter the pathname to the design component, or find and select the design using the navigator. Leave *default* for the viewpoint name, and **OK** the dialog box. The simulators use *default* as the default viewpoint name.

Choose the **Setup > (Quick)SIM, Fault, Path and Grade** pulldown menu item, shown in Figure 3-8, to create the default engineering (simulation) viewpoint. As with the PCB viewpoint, DVE adds the visible properties, and sets the primitive level in the viewpoint.
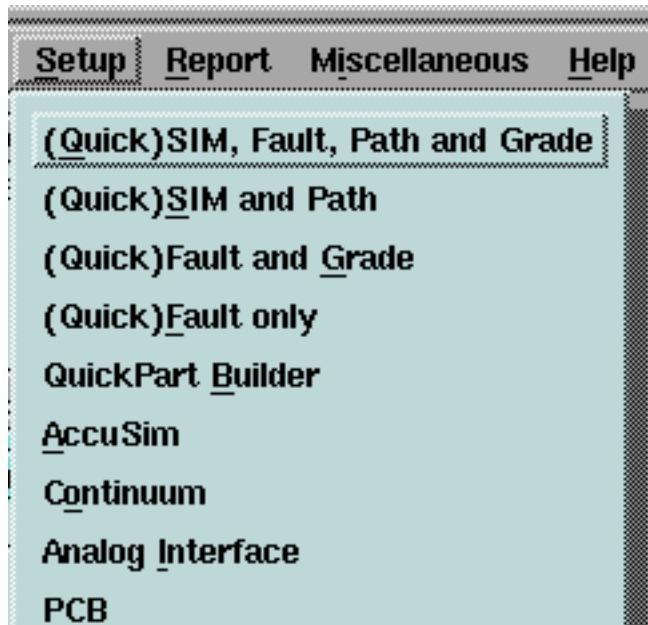
Figure 3-8. Setup > (Quick)SIM, Fault, Path and Grade Menu Item

Customize your viewpoint in the same way you customized the PCB
viewpoint:

- Use **Design Configuration > Add > Visible Property** to add any
  user-defined properties to the list of visible properties and to
  declare nets to be legal owners of propagated pin properties.

- Use **Palette > Add Param** to define values for any variables in
  property values that are needed for design evaluation.

- Use **Palette > Add Prim** to change the level to which the design is
  evaluated.

When you are finished editing the design viewpoint, click on the
Palette > Save Design icon.

Connecting Back Annotation Objects

PCB and simulation applications can share back annotation objects if they are properly connected to the design viewpoints. The modified and additional property values in a back annotation object overlay the design. A viewpoint can have more than one annotation object connected, which lets you see annotations from different applications or users. Your property edits are saved in the back annotation object having the highest priority. The most recently connected back annotation object always has the highest priority. Figure 3-9 illustrates one method of configuring viewpoints and annotation objects.
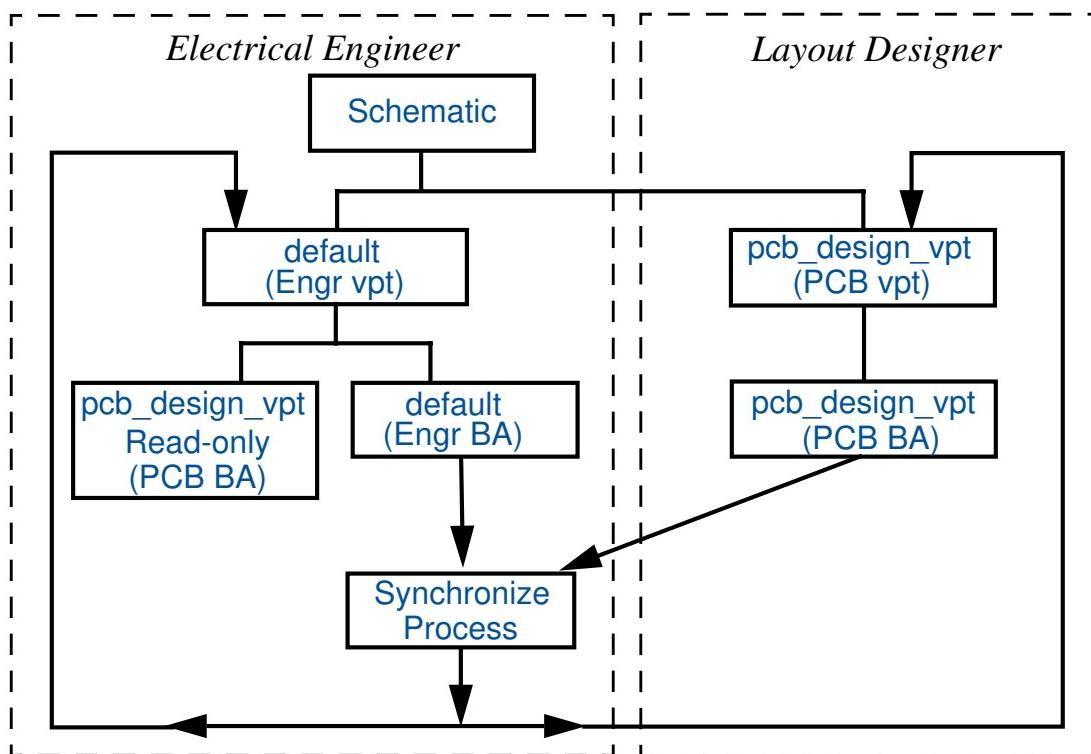

Figure 3-9. Viewpoint and Back Annotation Configuration

In this configuration, circuit design and simulation back annotations are written into *default*, which is connected to a simulation viewpoint. PCB back annotations are written into *pcb_design_vpt* which is connected to a PCB viewpoint.

As you can see in Figure 3-9, the PCB back annotation object is also connected to the engineering viewpoint, read-only. This is so the electrical engineer can see any design data back annotated by the layout designer, without locking the layout designer out of the PCB design viewpoint.

When you set up a design viewpoint for PCB, DVE automatically creates a back annotation object and connects it to the viewpoint. DVE does not automatically create and connect an engineering back annotation object.

The following steps show how to connect back annotation objects to the engineering viewpoint. You can use these steps to help you write scripts to create the viewpoints and connect the back annotation objects.

1. To open the engineering design viewpoint, click on the **Palette > Open Vpt** icon.

2. Use the navigator to specify the component name.

3. Leave *default* for the viewpoint name, and **OK** the dialog box.

   The Design Viewpoint window lists any connected back annotation objects and their priority. There should not be any connected to this viewpoint. If there are, disconnect them by choosing **File > Back Annotation > Disconnect** and filling in the dialog box.

4. To connect a PCB back annotation object, choose the **File > Back Annotation > Connect** pulldown menu item.

5. Select the viewpoint name, *pcb_design_vpt*, in the Navigator.

6. Click the **Read-only: Yes** button, and **OK** the dialog box, as shown in Figure 3-10.
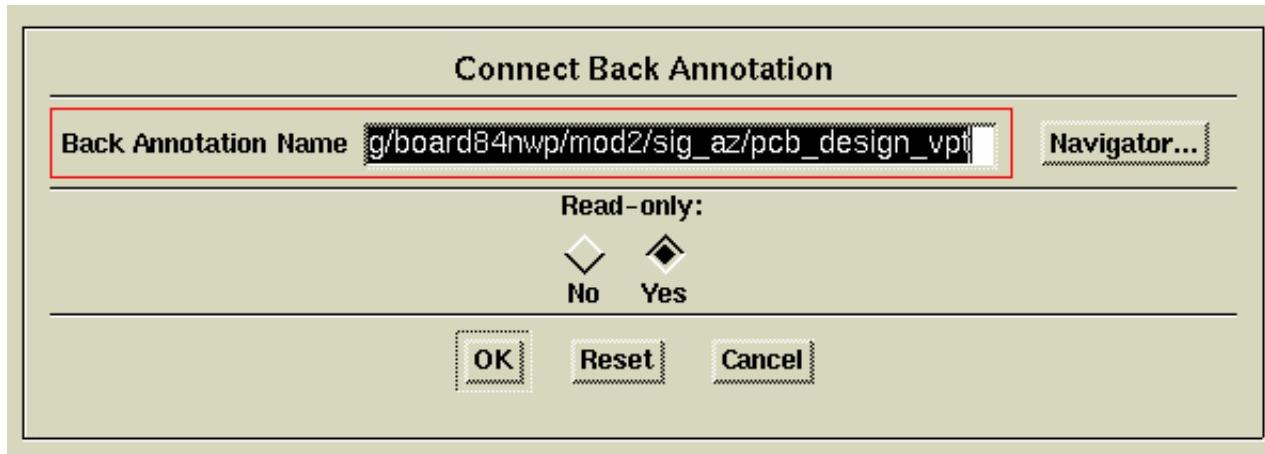
Figure 3-10. Connect Back Annotation Dialog Box

7. To create a new back annotation object for electrical engineering edits, choose **File > Open > Back Annotation**.

8. In the dialog box shown in Figure 3-11, enter *default*, and click **OK**.
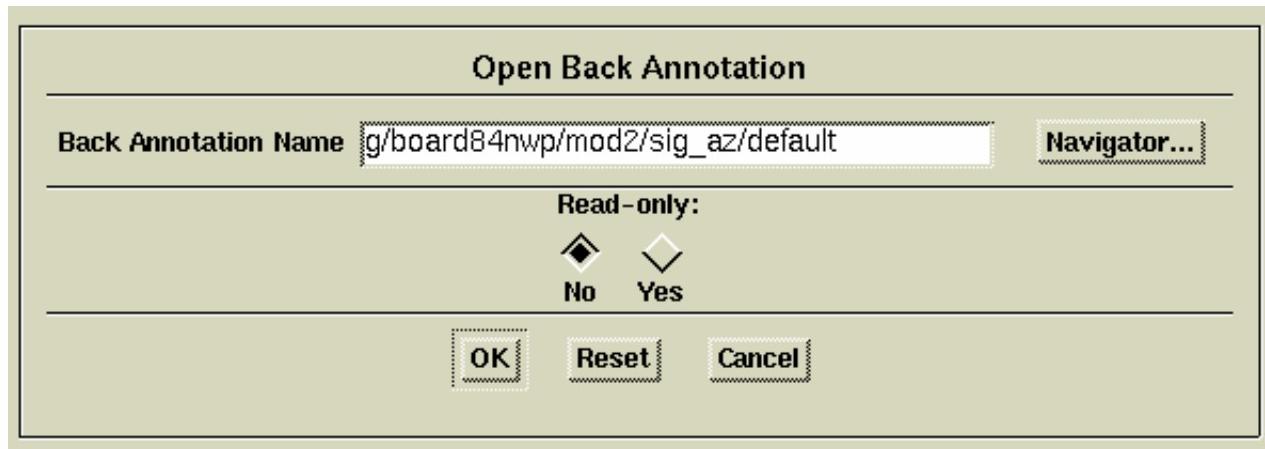
Figure 3-11. Open Back Annotation Dialog Box

Figure 3-12 shows the Design Viewpoint window.
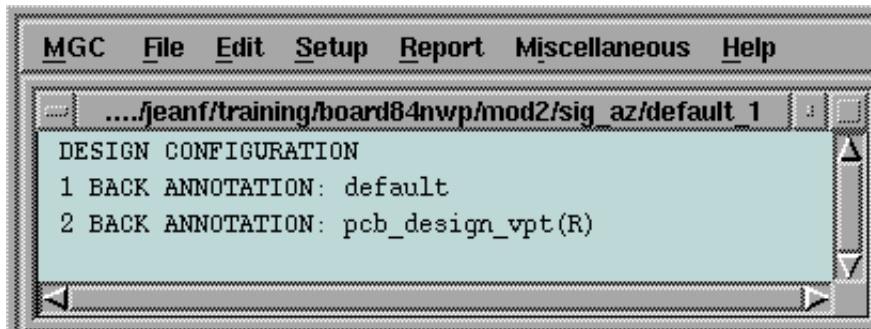
Figure 3-12. Back Annotations Connected to Engineering Viewpoint

This shows that property edits will be written to the simulation back annotation object when the design is seen through the engineering design viewpoint. The *pcb_design_vpt (R)* indicates you will be able to see the PCB back annotations, but cannot write edits there.
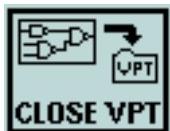
9. To save and close the viewpoint, click on **Palette > Save Design**, then on **Palette > Close Vpt**.

Checking the Design in DVE

In Design Architect, you checked single schematic sheets and multiple schematic sheets at the same level. Now you need to check the entire design hierarchy using the design configuration rules defined in the design viewpoint. DVE examines the design for such things as mismatched connections, unique names, and parameter values. Figure 3-13 illustrates design checking in DVE.
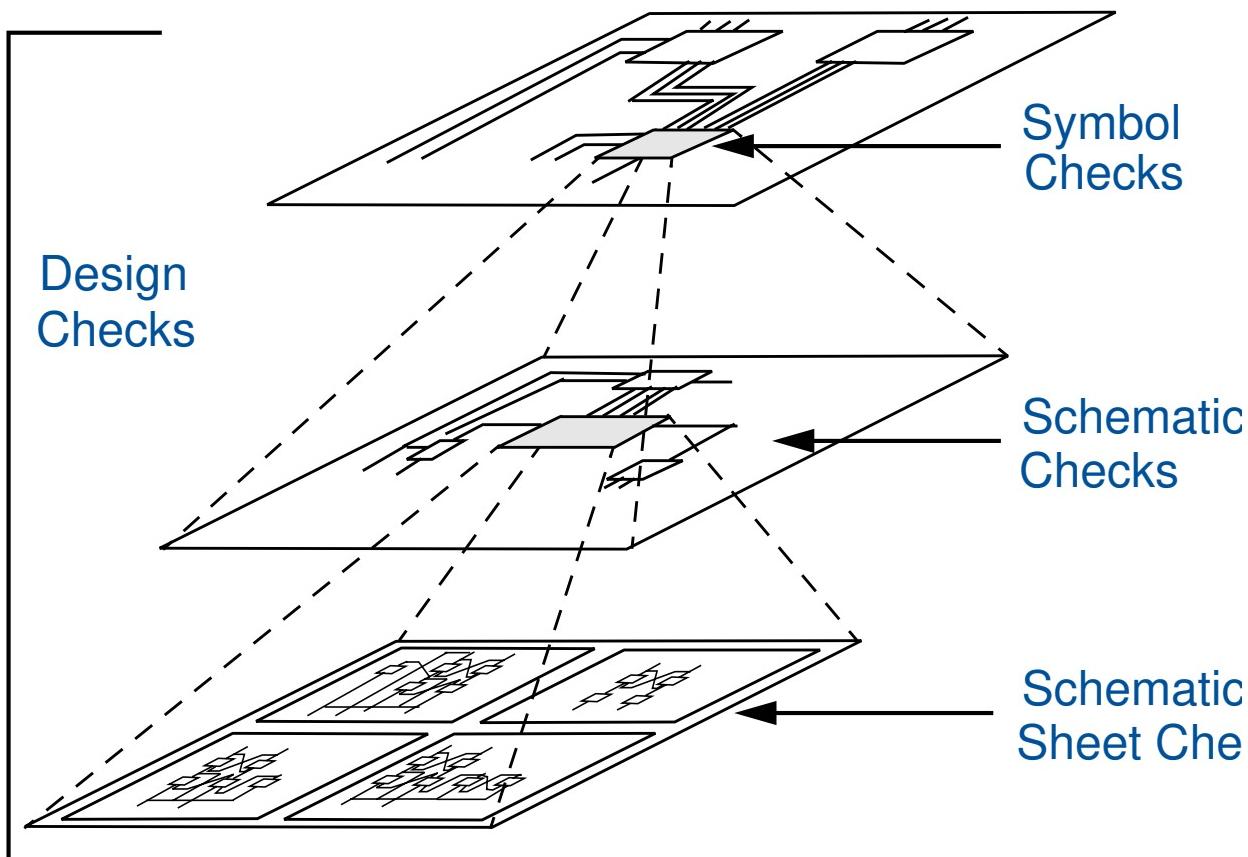

Figure 3-13. Design Checking

To check the design hierarchy in DVE, follow these steps:

1. You need to open the design viewpoint you wish to use, and then choose the **Miscellaneous > Check Design > Basic Checks** pulldown menu item.

   DVE checks all sheets in the design, then displays the results in a message window.

2. To locate objects that caused errors or warnings, select a message in the message window and choose **File > Open > Selected**.

   This opens a Schematic View window, or a VHDL View window if the error is in a VHDL model, centering the selected problem area.

   Generally, you can correct errors in one of three ways:

   - Editing schematic sheets, symbols, or VHDL models in Design Architect.

   - Adjusting design configuration rules in DVE.

   - Adding or editing back annotations in DVE.

   If you need to make corrections in Design Architect, leave DVE open.

3. Invoke Design Architect, make your changes, check the sheet, and save the component.

4. To re-check in DVE, choose the **Edit > Reload Model** menu item and specify in the dialog box which models should be reloaded.

5. Repeat this correction loop until there are no errors.

For more information about viewpoints, refer to the *Design Viewpoint Editor User's and Reference Manual*.

Lab Exercise

In this lab exercise, you create two design viewpoints, and connect two back annotation objects. You look at the hierarchy of the design, and check the design in DVE. Upon completion of this lab exercise, you should be able to:

- Invoke the Design Viewpoint Editor (DVE).

- Set up a PCB design viewpoint.

- Set up a simulation design viewpoint.

- Connect back annotation objects to each viewpoint.

- Check a design in DVE.

Turn to Module 2—Lab 3: "Creating Design Viewpoints".

Lab 3
Creating Design Viewpoints

Introduction

In this lab exercise, you create two design viewpoints, and create and connect two back annotation objects. You look at the hierarchy of the design, and check the design in DVE. Upon completion of this lab exercise, you should be able to:

● Invoke the Design Viewpoint Editor (DVE).

● Create and set up a PCB design viewpoint.

● Create and set up an engineering design viewpoint.

● Check a design in DVE.

Procedure

Open DVE to create and set up your viewpoints.

Creating a PCB Design Viewpoint

1. Log in to your workstation.

2. Invoke the Design Manager from a shell:

      `$MGC_HOME/bin/dmgr`

3. Invoke DVE by double-clicking the Select mouse button on the DVE icon in the Tools window.

DVE opens in a new window.

4. Click the Select mouse button on the **Palette > Open Vpt** icon.

The Open Design Viewpoint dialog box is displayed, prompting you for a component name.

5. Click the Select mouse button on the **Navigator** button. When the Navigator is displayed, use the **Goto** button to navigate to the lab exercise design. The pathname is:

   your_path**/training/board_new/mod2/sig_az**

6. Enter **pcb_design_vpt** in the Viewpoint Name text entry box, and **OK** the dialog box.

You could leave "default" in the text entry box, and DVE would automatically change the name to "pcb_design_vpt" when you save the viewpoint. DVE opens a new, empty viewpoint, and displays a Design Viewpoint window and a Design Configuration window, as shown in Figure 3-14.
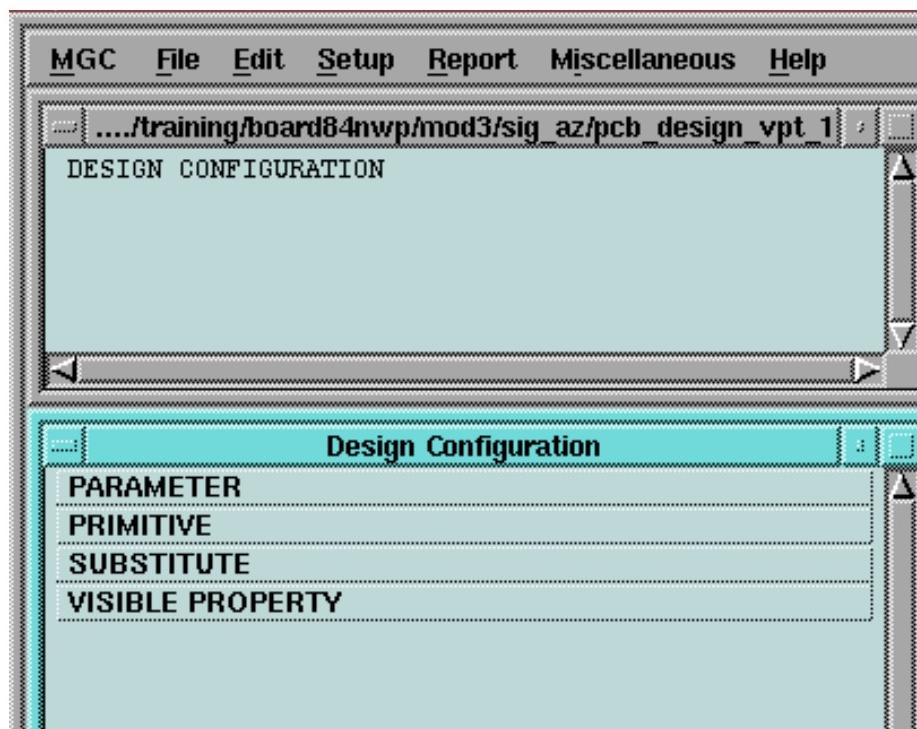
Figure 3-14. Empty Viewpoint Windows

7. Choose the **Setup > PCB** pulldown menu item shown in Figure 3-15.
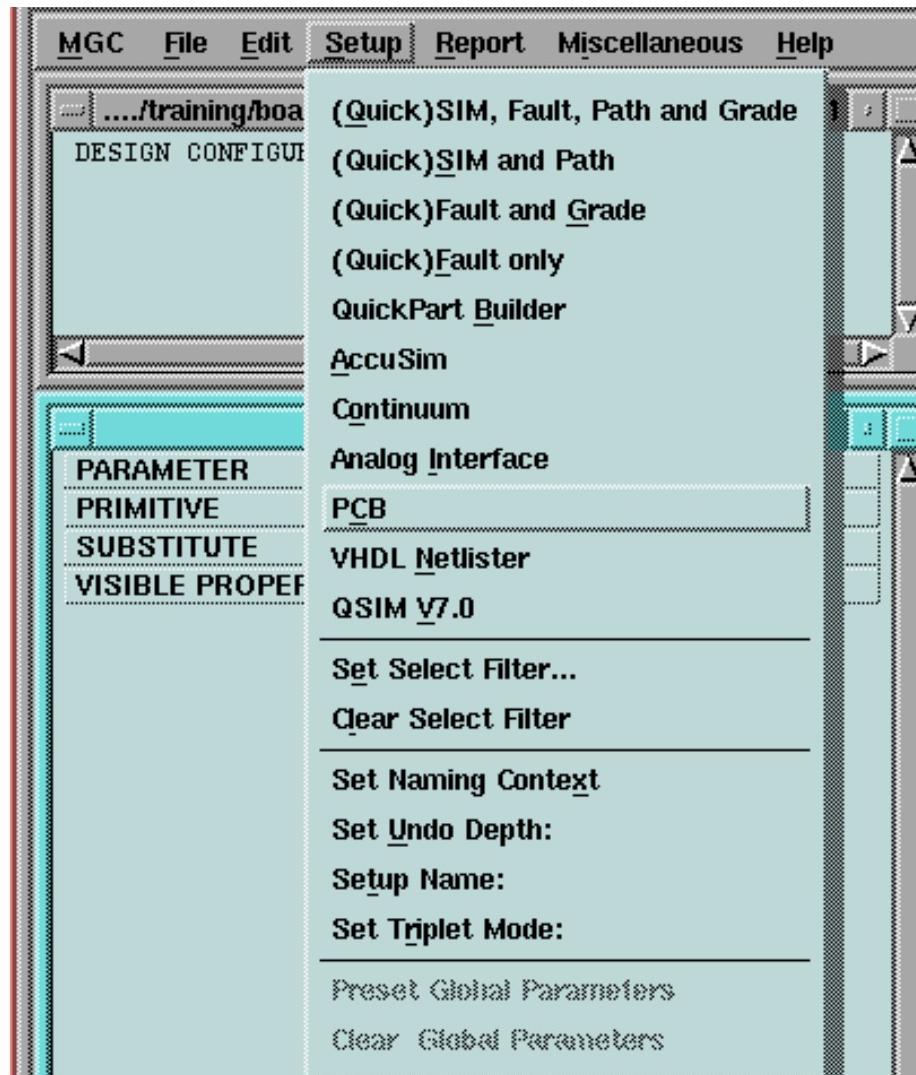
Figure 3-15. Setup > PCB Menu Item

All properties needed by the Board Station tools are placed on the visible property list, and all components having the Comp property are considered primitive. A back annotation object, **pcb_design_vpt**, is automatically created and connected to the design viewpoint for future editing from PCB tools. Connected back annotation objects are listed in the Design Configuration window, as shown in Figure 3-16.
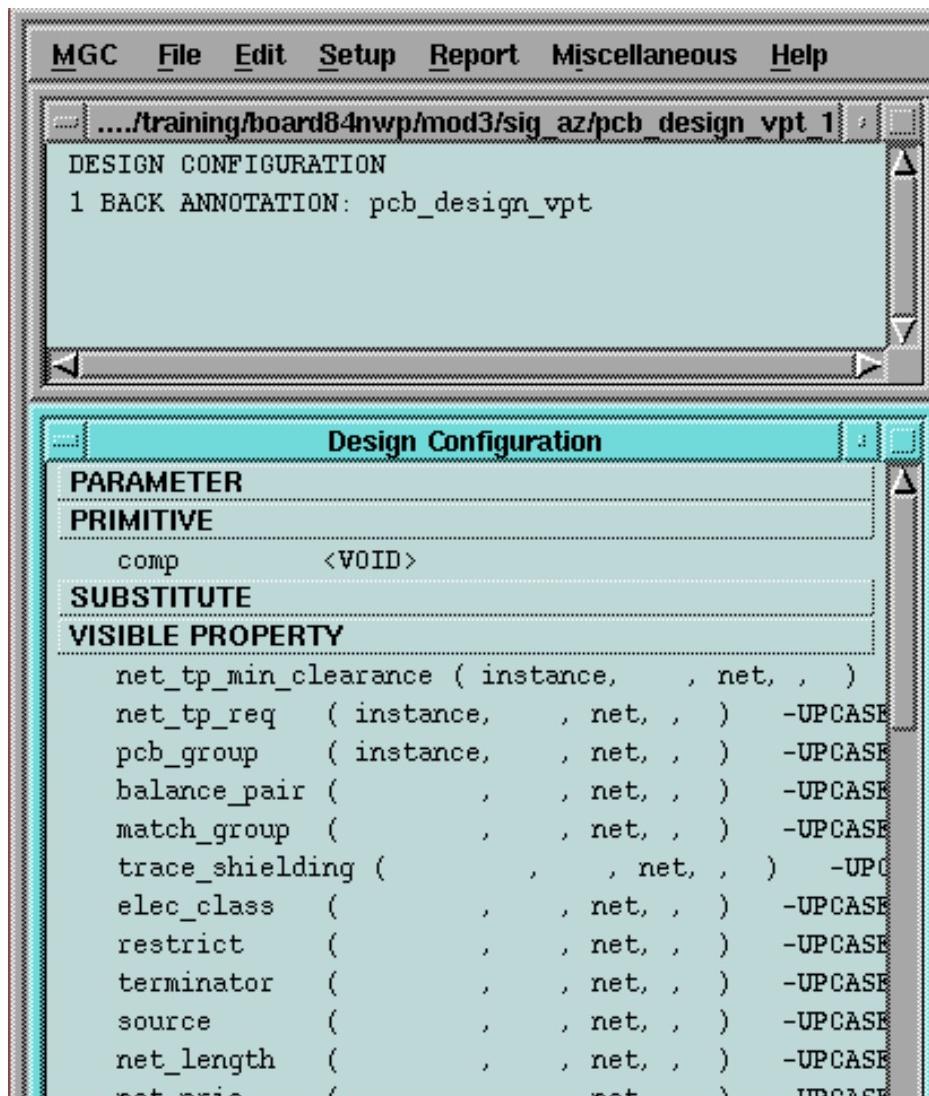
Figure 3-16. Design Viewpoint and Configuration Windows After Setup

When you edit properties from a PCB tool, those edits are written
in the **pcb_design_vpt** back annotation object. The visible
property list and the primitive list are part of the viewpoint, and are
not written in a back annotation object.

This design contain frames that have a frame expression
"IF LAYOUT == 1". This means that the circuitry within a frame
having this expression is only used by PCB tools. If the variable
LAYOUT has any other value, the circuitry within the frame is
ignored. This design viewpoint is for Board Station tools, so the
value of LAYOUT should beset to "1".

For information about creating frames on a schematic sheet, refer to
the *Design Architect User's Manual*.

8. Click the Select mouse button on the **Palette > Add Param** icon.

   The Add Parameter dialog box is displayed.

9. Complete the dialog box as shown in Figure 3-17, and click the **OK**
   button.
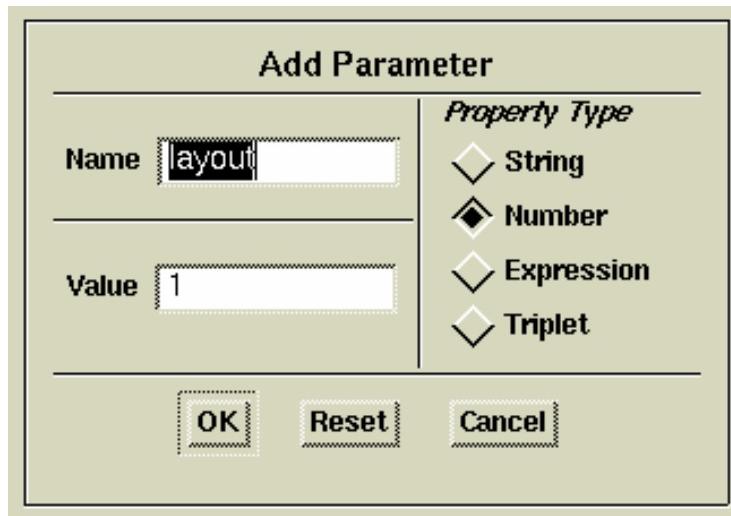
Figure 3-17. Add Parameter Dialog Box

The parameter appears in the Design Configuration window, shown
in Figure 3-18.
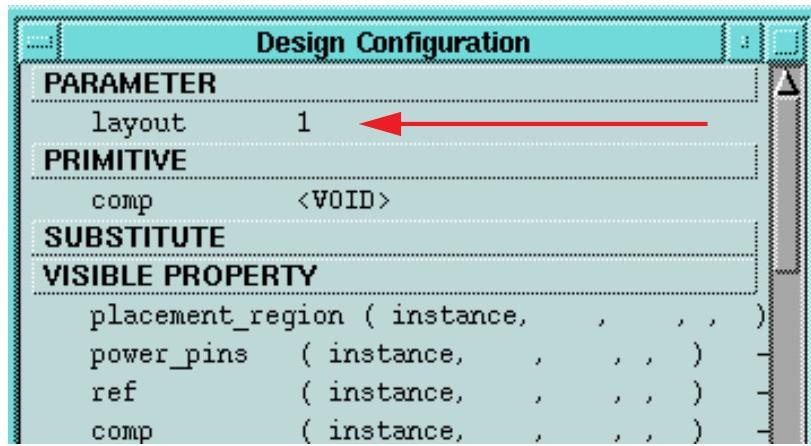
Figure 3-18. Design Configuration Window

Checking and Saving the PCB Viewpoint

Although you do not have to run DVE checks, it is recommended that you check your design against the primitive level and parameters set in the design viewpoint. This checking ensures that downstream applications can use the design, and that the hierarchy is connected correctly and is complete.

1. Choose the **Miscellaneous > Check Design > Basic Checks** pulldown menu item, shown in Figure 3-19.
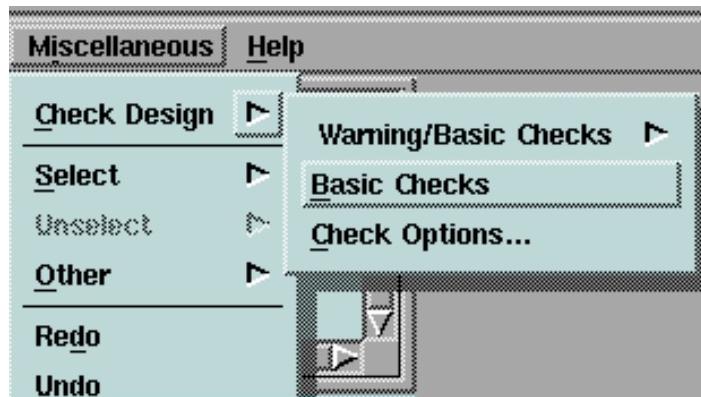

Figure 3-19. Miscellaneous > Check Design Menu

Errors are listed in a Design Syntax Messages window similar to the one in Figure 3-20.

Design Syntax Messages

Design Syntax Checker - viewpoint $MGC_HOME/shared/training/board84n
Performing the following categories of checks:
  Basic syntax checks

Note: Design Syntax Checks completed - 0 status messages returned. (
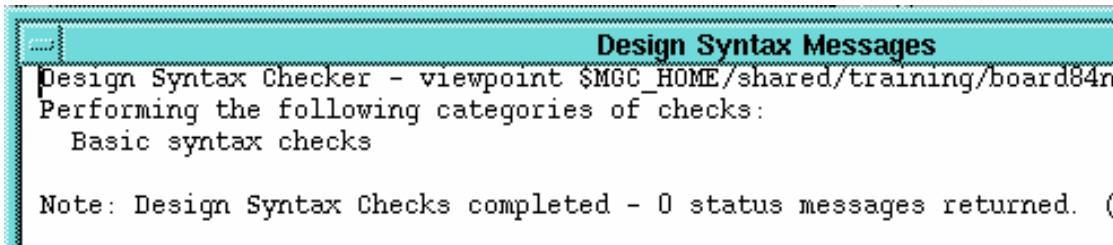
Figure 3-20. Design Syntax Messages Window

2. Close the Design Syntax Messages window.

3. Click the Select mouse button on the **Palette > Save Design** icon to save the viewpoint information.
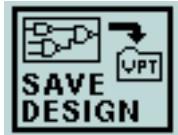
The design viewpoint is saved and remains open.

Looking through the PCB Viewpoint

Now that the PCB design viewpoint is created, you can see how the design appears to Board Station tools. You will open a few sheets of the design and observe the interaction between the sheets as you select objects.

1. Click the Select mouse button on the **Palette > Open Sheet** icon.

The Open Sheet dialog box is displayed, prompting you to choose which sheet(s) to open.

2. Select sheets 1 and 2 listed in the dialog box by pressing and holding the Select mouse button down as you drag the cursor over both names. **OK** the dialog box.

DVE opens both sheets in separate Schematic windows.

3. Select the **DATA_IO** component on **sheet1**, then choose the **File > Open > Down** menu item shown in Figure 3-21.

The schematic for the DATA_IO component is displayed in a new window.
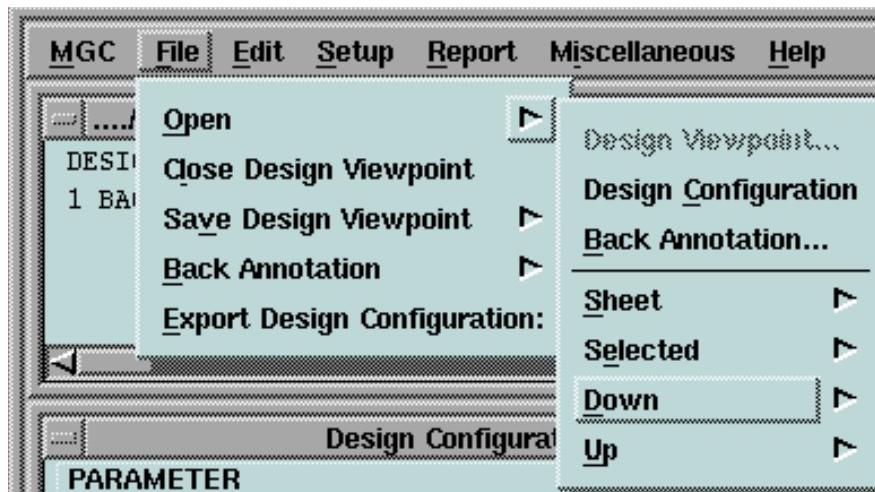
Figure 3-21. File > Open Menu

4. Select the HI_SPEED_CHANNEL component on the DATA_IO sheet just opened, and choose the **File > Open > Down** menu item.

You now have four schematic windows open and stacked on top of each other. Next, you will rearrange the windows.

5. Choose the **MGC > Setup > Session** pulldown menu item shown in Figure 3-22.
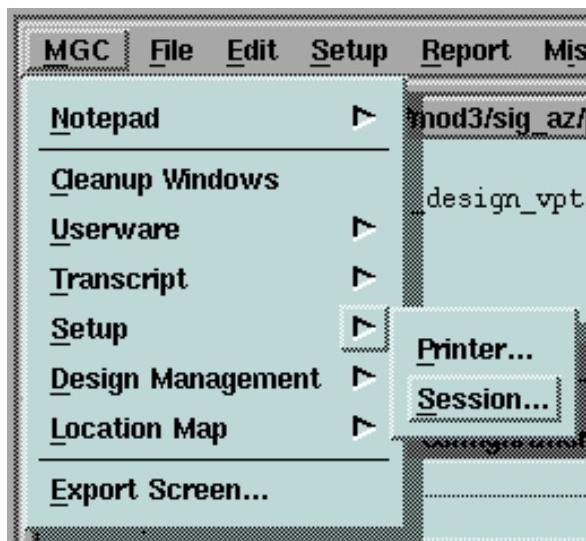

Figure 3-22. MGC > Setup Menu

The Setup Session dialog box, shown in Figure 3-23, is displayed for you to choose an input device, double-click speed, window area visibility, and window layout.

Setup Session

Select Input Device:

X11_POINTER

☑ Show Menu Bar

☐ Show Session Title

☑ Show Message Area

☑ Show Softkey Area

Double Click Speed

◇ Slow
◇
◆
◇
◇ Fast

Window Layout

◇ Stacking
◇ Up Down Tiling
◆ Quadrant Tiling
◇ Left Right Tiling
◇ Ask User for Position

OK   Cancel
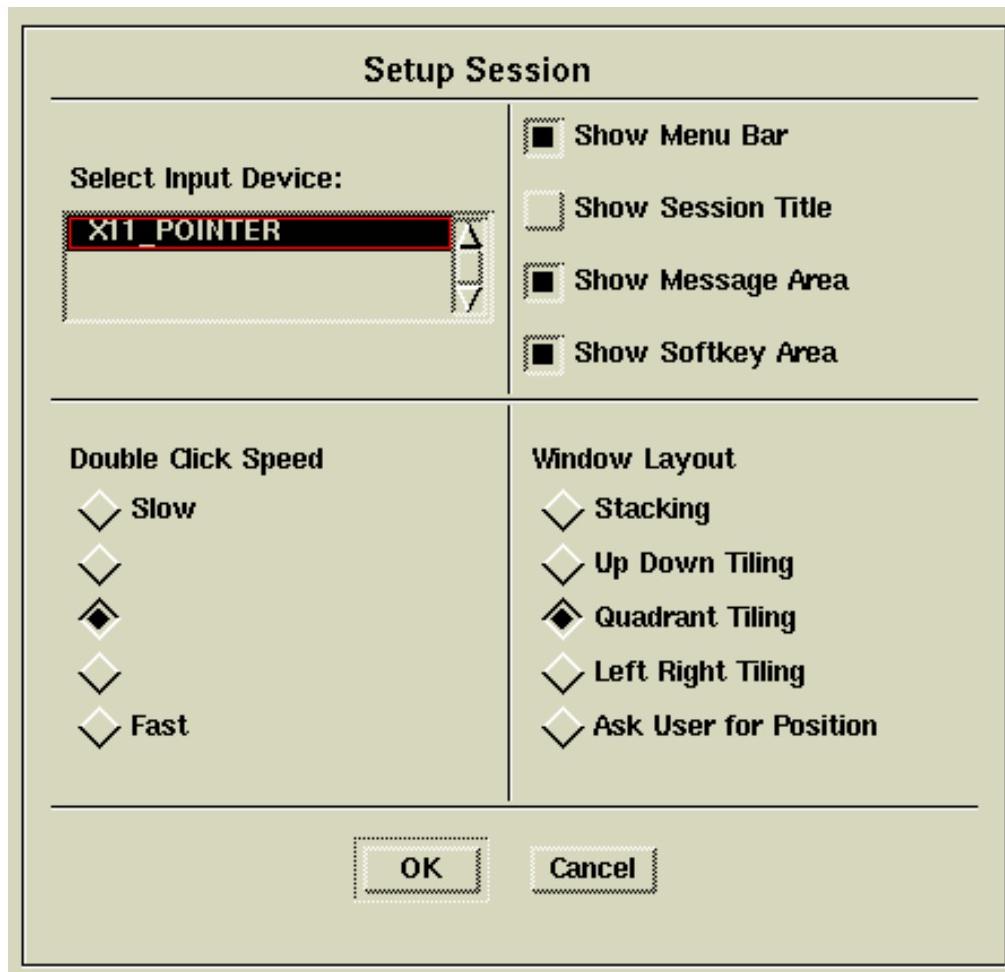
Figure 3-23. Setup Session Dialog Box

6. Choose **Window Layout: Quadrant Tiling**, and **OK** the dialog box.

The Design Viewpoint window, Design Configuration window, and the four schematic windows are redrawn. Locate all four schematic windows using the **Window > Pop** menu item, if necessary. You may need to move one or more windows to a different quadrant in order to see all the schematic windows.

7. Select the circuitry on the left side of **sheet2**.

Notice that connected objects on the HI_SPEED_CHANNEL sheet are also selected.

8. Select some buses, one at a time, on the **DATA_IO** sheet.

Notice that the same objects are selected on other sheets. When you select a bus, some wires appear aquamarine instead of white. Aquamarine is used to show *containment*; if you select a bus, you can see where the individual wires contained in that bus are routed.

9. Press the Unselect All function key.

All objects on all open sheets are unselected.

10. Click on the **Palette > Close Vpt** icon.

You already saved the viewpoint earlier, so you do not need to save it now. Closing the viewpoint closes all windows.

Creating an Engineering Design Viewpoint

1. Click the Select mouse button on the **Palette > Open Vpt** icon.

The Open Design Viewpoint dialog box is displayed with the component name used previously:

your_path/**training/board_new/mod2/sig_az**

2. Enter **default** for the Viewpoint Name, and **OK** the dialog box.

DVE opens a new, empty viewpoint.

3. Choose the **Setup > (Quick)SIM, Fault, Path and Grade** pulldown menu item, as shown in Figure 3-24.
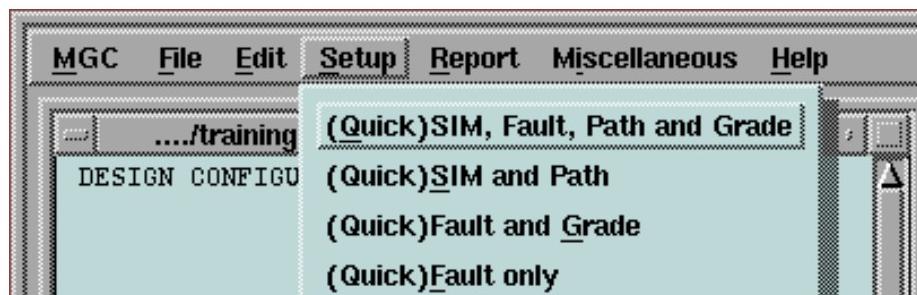
Figure 3-24. Setup > (Quick)SIM, Fault, Path and Grade Menu Item

All properties used by the simulation tools are made visible. DVE
also sets the correct primitive level for simulation, using the value
of the Model property. The Design Configuration window is shown
in Figure 3-25
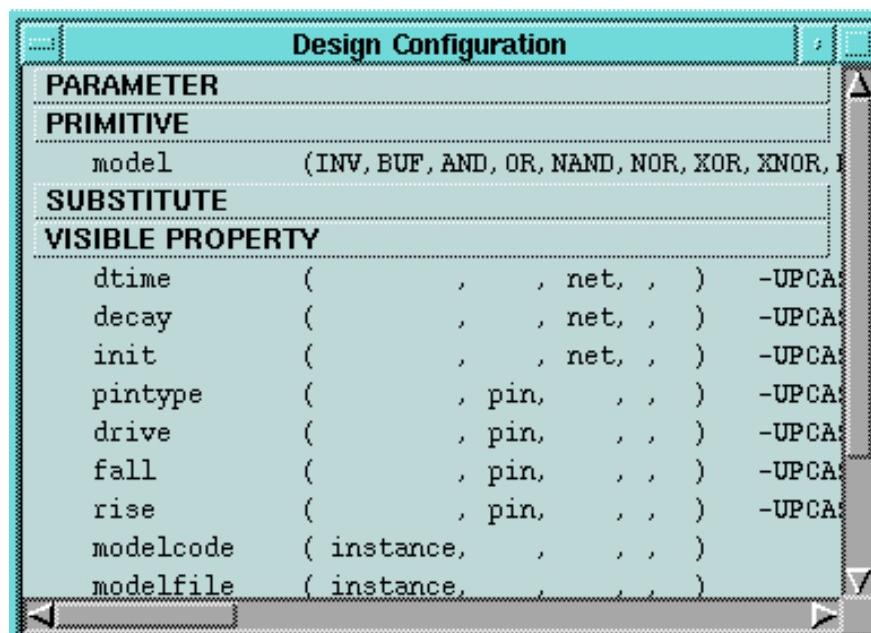
Figure 3-25. Design Configuration Window

The frame expression "IF LAYOUT == 1" needs a value for the
variable LAYOUT so the design can be evaluated. Because this
viewpoint affects simulation and not board layout, you will set the
variable LAYOUT to be 0. For this design, the result is the same if
you do not set this parameter; however, this may not be true for all
designs, so you should set values for your variables.

4. Click the Select mouse button on the **Palette > Add Param** icon.

The Add Parameter dialog box is displayed.

5. Complete the dialog box as shown in Figure 3-26, and click the **OK** button.
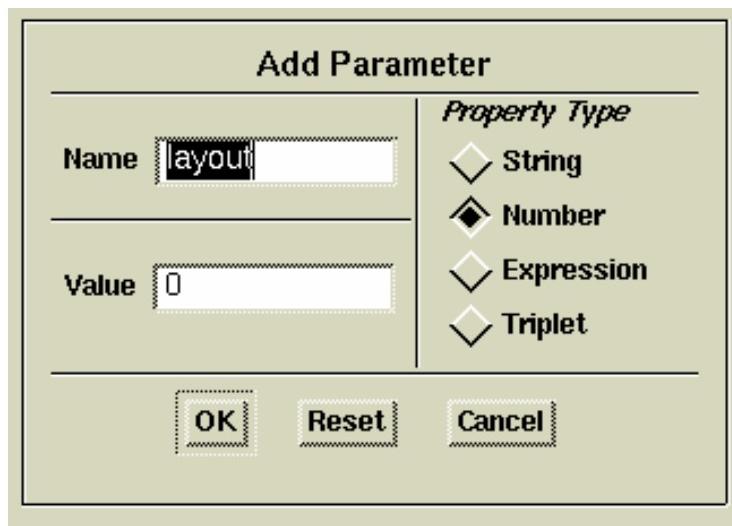
Figure 3-26. Add Parameter Dialog Box for Engineering Viewpoint

The Design Configuration window displays the new parameter value, as shown in Figure 3-27.
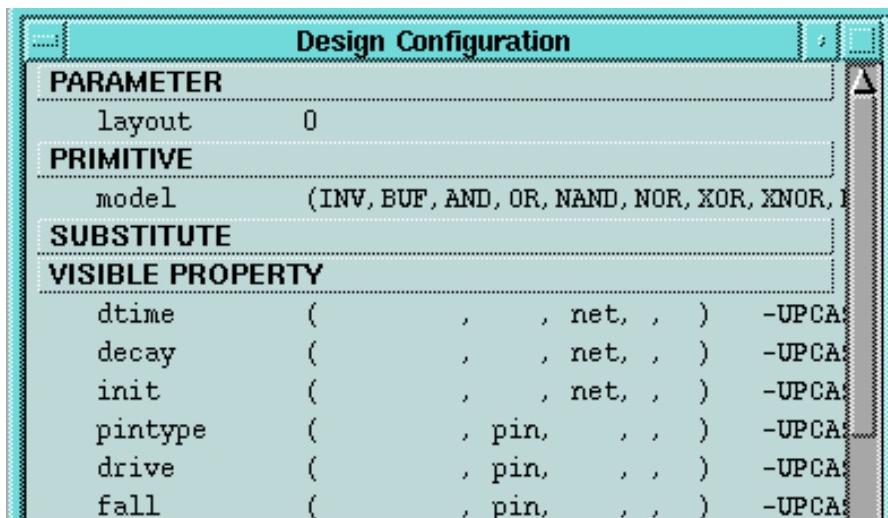
Figure 3-27. Design Configuration Window for Engineering Viewpoint

Checking and Saving the Engineering Viewpoint

1. Choose the **Miscellaneous > Check Design > Basic Checks** pulldown menu item.

   The design is checked against the primitive level and parameters you set in the engineering viewpoint. Error messages are listed in a Design Syntax Messages window. You will get some errors concerning pins without matching nets on the a_d_block component. You will look at the design to find the problem.

2. Close the Messages window.

3. Click the Select mouse button on the **Palette > Open Sheet** icon.

4. In the Open Sheet dialog box, click the Select mouse button on **sheet1** and **OK** the dialog box.

5. Select the **DATA_IO** component, then choose the **File > Open > Down** menu item. Figure 3-28 shows where the **DATA_IO** component is located.
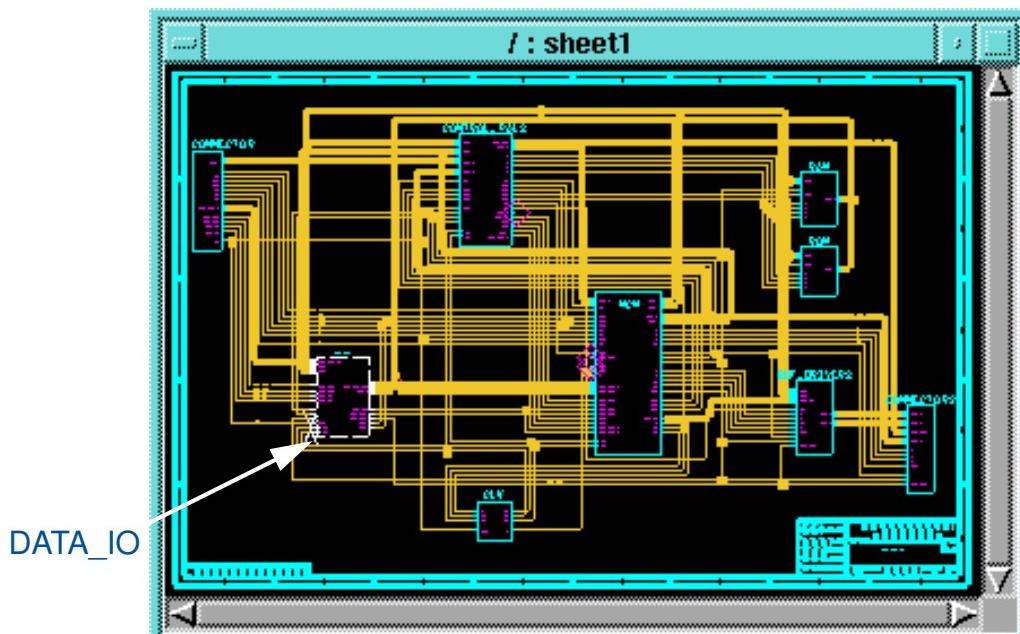

Figure 3-28. DATA_IO Component Location

6. Click the Select mouse button on the **a_d_block** component on the **DATA_IO** sheet, then choose the **File > Open > Down** menu item.

DVE displays the sheet representing the a_d_block. Do you see the reason for the design checking errors?

The entire sheet is enclosed in a frame, and DVE was trying to match pins to underlying nets which do not exist in this viewpoint because the variable LAYOUT is set to "0". For the purpose of simulation, the a_d_block component should be primitive.
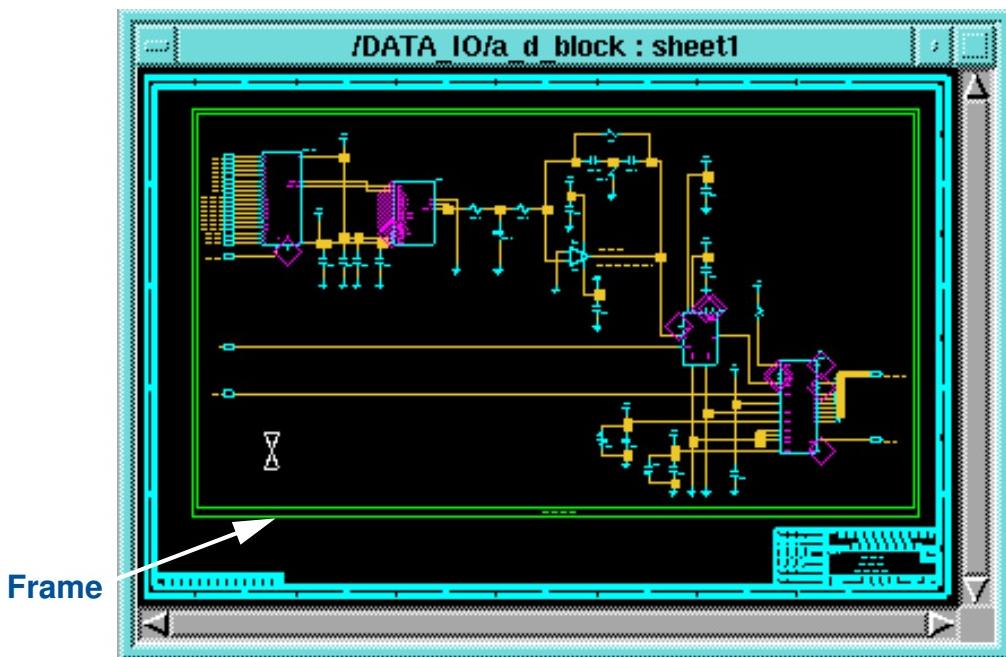

Figure 3-29. Frame on DATA_IO Sheet

7. Choose the **Report > Objects > Long** pulldown menu item.

The report, shown in Figure 3-30, displays the values of properties assigned to the a_d_block component.

8. Find and write down the Inst property value for the next steps, then close the Report window.

The simulators use the Model property to define which components are primitive. You will assign the Model property the same value as the Inst property, and then add the value to the primitive list.
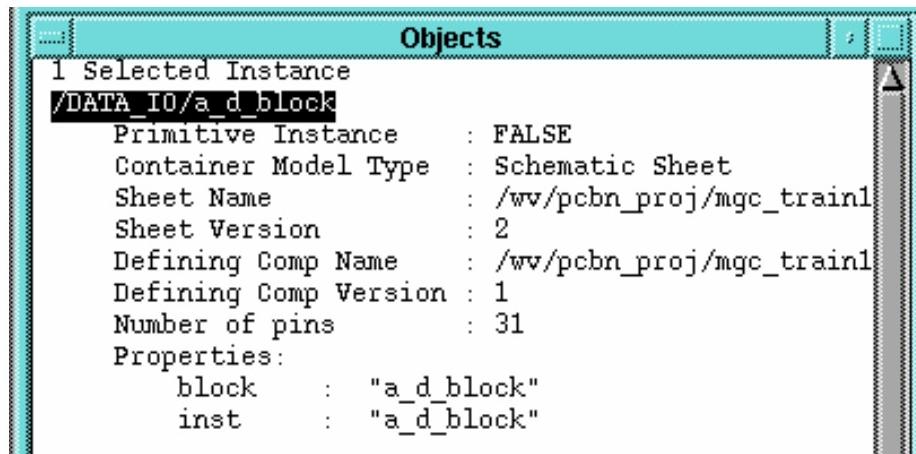
Figure 3-30. DATA_IO Report
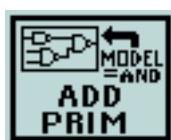
9. Click the Select mouse button on the **Palette > Add Prim** icon.

The Add Primitive dialog box is displayed.

**10.** Complete the Add Primitive dialog box as shown in Figure 3-31, then click the **OK** button.
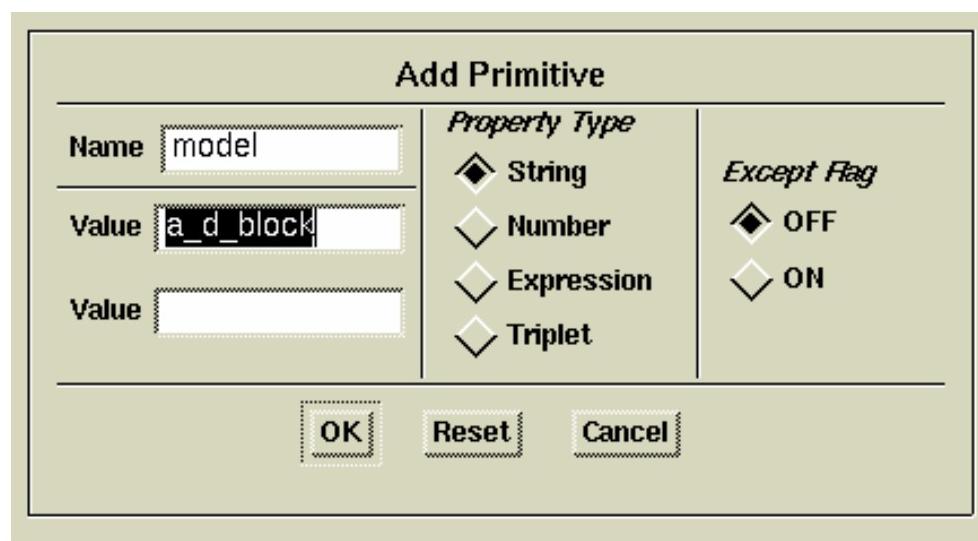
Figure 3-31. Add Primitive Dialog Box

In the Design Configuration window, "a_d_block" is added to the primitive list, as shown in Figure 3-32.
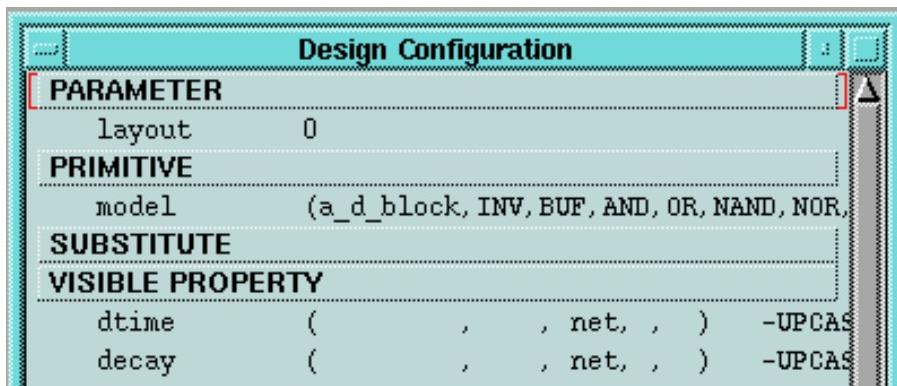
Figure 3-32. a_d_block Added to the Primitive List

DVE redraws each of the displayed sheets, except the a_d_block sheet. Figure 3-33 shows the Messages window informing you that the view of the a_d_block sheet was deleted. The sheet is still part of the design, it is not seen by the engineering viewpoint because you specified that a_d_block is primitive.
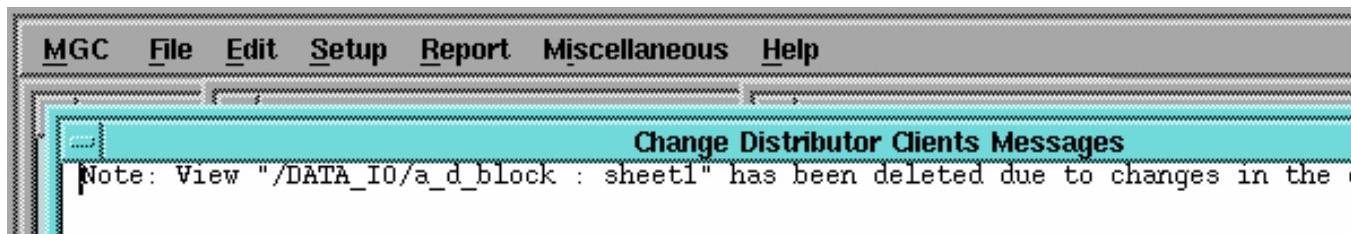

Figure 3-33. View of a_d_block Sheet Deleted from Viewpoint

11. Close the Messages window.

12. Choose **Miscellaneous > Check Design > Basic Checks** to check the design again.

There should be no errors reported now.

13. Close the Design Syntax Messages window.

14. Click the Select mouse button on the **Palette > Save Design** icon to save the viewpoint information.

The design viewpoint, **default**, is saved and remains open.

Looking through the Engineering Viewpoint

Now you will open the same sheets that you opened with the PCB viewpoint so you can look at similarities and differences in the two viewpoints.

1. Select the **HI_SPEED_CHANNEL** component on the **DATA_IO** sheet just opened, then choose the **File > Open > Down** menu item.

2. Activate the edit window in which **sheet1** of **sig_az** is displayed (the window banner shows "/:sheet1"), and click the Select mouse button on the **Palette > Open Sheet** icon.

3. Select **sheet2** from the dialog box and click the **OK** button.

4. Rearrange windows, as necessary, to see all schematic windows.

5. Select the circuitry on the left side of **sheet2**.

   Can you select any objects on this sheet? Why, or why not?

   While looking through the engineering viewpoint, you cannot select any objects within a frame having the expression "IF LAYOUT == 1" because you assigned a value of "0" to the variable LAYOUT. Unless the value is "1", this viewpoint does not know the frame exists.

6. Select buses on **sig_az/sheet1** as you did with the PCB viewpoint, and see the results on the other sheets. When you are done, press the Unselect All function key.

7. Select the **BUF_DRIVERS** component on **sheet1** and choose the **File > Open > Down** menu item. Figure 3-34 shows the location of the BUF_DRIVERS component.
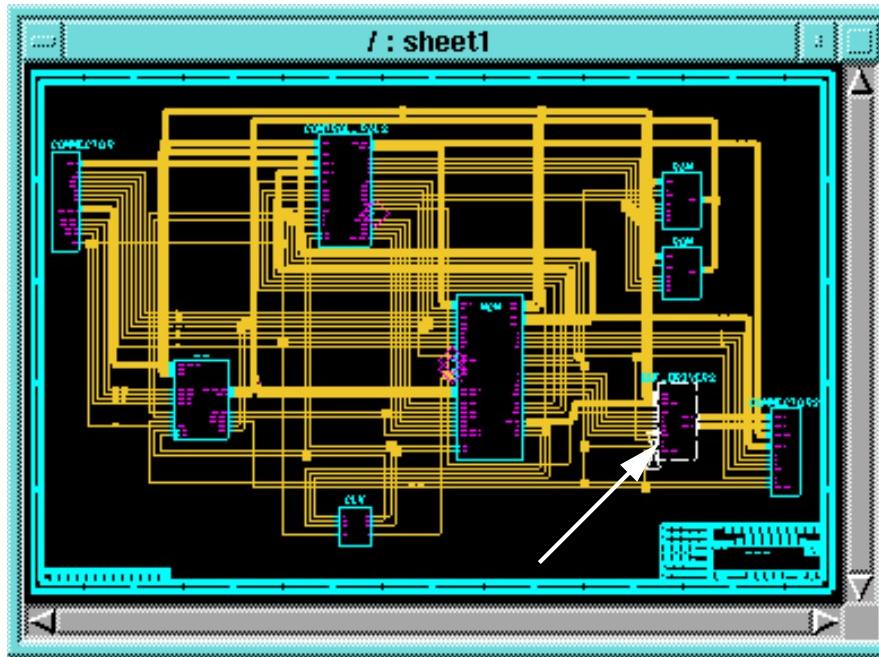
Figure 3-34. BUF_DRIVERS Component Location

8. Activate the newly opened BUF_DRIVERS sheet and choose the **File > Open > Up** menu item.

If the parent sheet is already open and hidden under another window, it is popped to the top of the display. If the parent sheet is open but not hidden, it is activated. If the parent sheet is not displayed, or if you chose **File > Open > Up > New**, DVE opens a new window to display that sheet.

9. Close all schematic windows using each window's **Window > Close** menu item, shown at left.

Connecting Back Annotation Objects

You now have two design viewpoints, **pcb_design_vpt** (closed) and **default** (open). The PCB design viewpoint has a back annotation object that was connected automatically.

You are going to connect two back annotation objects to the engineering design viewpoint and correctly prioritize them. You will connect the **pcb_design_vpt** back annotation object in read-only mode so you can see changes made by the layout designer. You will connect the **default** back annotation object in which you can write engineering edits.

1. Click the Select mouse button on the **Palette > Connect BA** icon.

   The Connect Back Annotation dialog box is displayed, prompting you for the pathname to the back annotation object you want to connect.

2. Click the **Navigator** button and choose the **pcb_design_vpt** back annotation object (with the "b" in the icon) from the list.

   The Navigator is shown in Figure 3-35. The design viewpoint pathname should be:

   your_path/**training/board_new/mod2/sig_az/pcb_design_vpt**
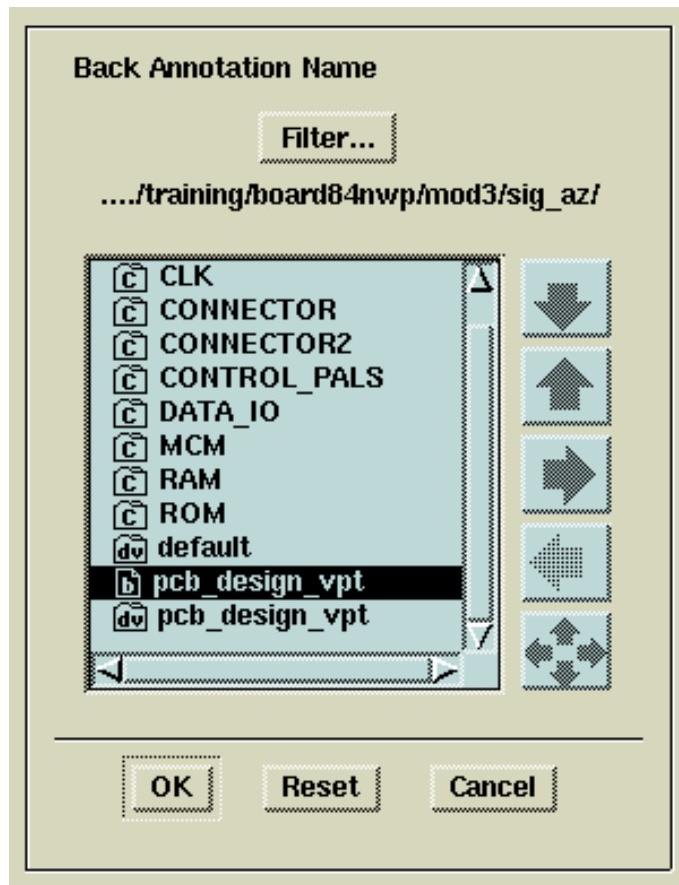
Figure 3-35. Navigator Box Listing pcb_design_vpt Back Annotation

        3. OK the Navigator; the pathname is automatically entered in the dialog box, shown in Figure 3-36. Click on **Read-only: Yes**, then **OK** the dialog box.

Connect Back Annotation

Back Annotation Name |g/board84nwp/mod2/sig_az/pcb_design_vpt| Navigator...

Read-only:

◇   ◆
No   Yes

OK   Reset   Cancel
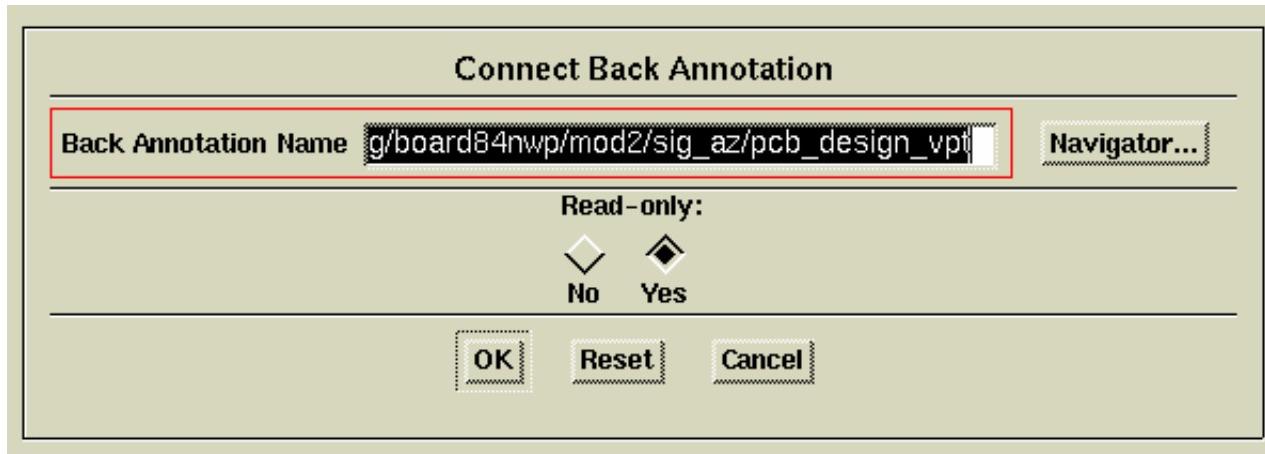
Figure 3-36. Connect Back Annotation Dialog Box

DVE connects the PCB back annotation object read-only to the engineering viewpoint, and displays this information in the Design Configuration window, shown in Figure 3-37.

MGC   File   Edit   Setup   Report   Miscellaneous   Help

..../jeanf/training/board84nwp/mod2/sig_az/default_1

```
DESIGN CONFIGURATION
1 BACK ANNOTATION: pcb_design_vpt(R)
```
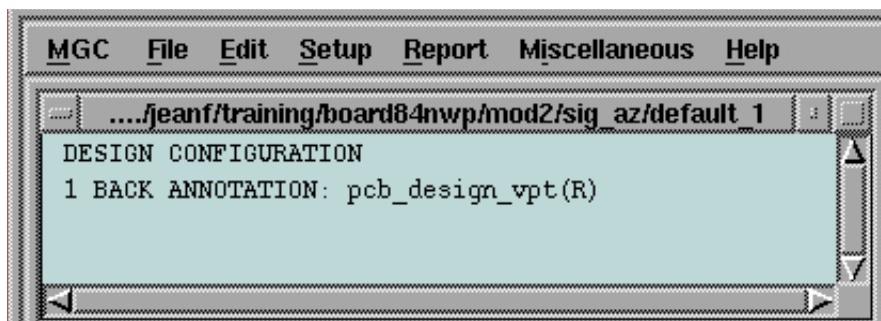
Figure 3-37. Back Annotation Connected to Engineering Viewpoint

Now you will open a new back annotation object for future schematic and simulation edits.

4. Choose the **File > Open > Back Annotation** pulldown menu item shown in Figure 3-38.
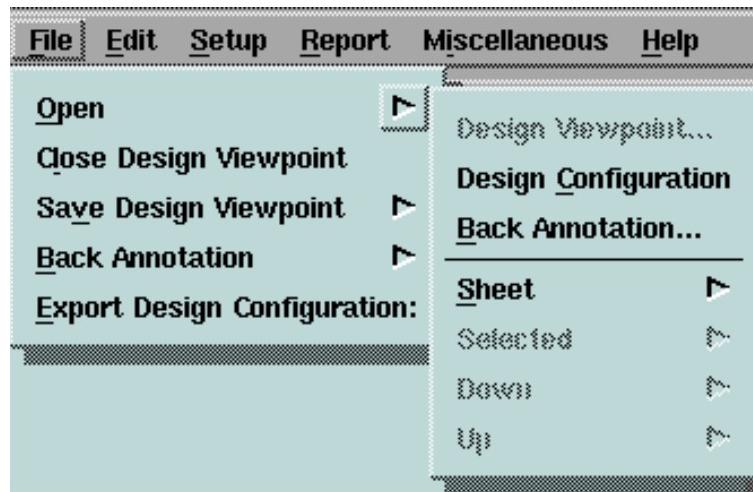
Figure 3-38. File > Open Menu

5. Enter the pathname for a back annotation object named **default** in the dialog box and be sure **Read only: No** is specified, as shown in Figure 3-39.
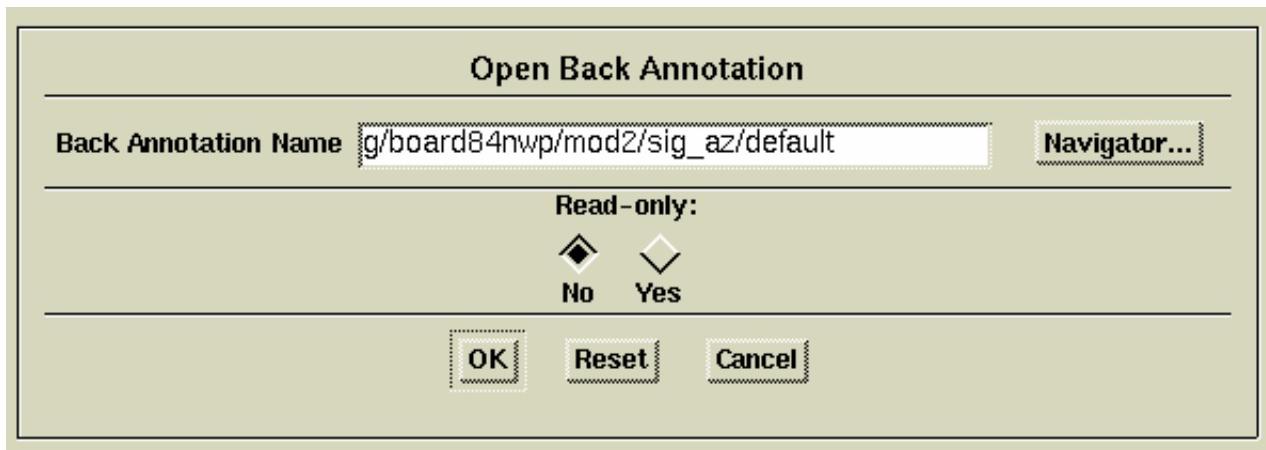

Figure 3-39. Open Back Annotation Dialog Box

Now the Design Viewpoint window shows both back annotation objects connected, with **default** having the highest priority (1), and **pcb_design_vpt** having a lower priority (2) and connected read-only. The updated Design Viewpoint window is shown in Figure 3-40.

```
MGC   File   Edit   Setup   Report   Miscellaneous   Help

 ─|   ..../jeanf/training/board84nwp/mod2/sig_az/default_1   |·||□|
  DESIGN CONFIGURATION                                          △
  1 BACK ANNOTATION: default
  2 BACK ANNOTATION: pcb_design_vpt(R)
                                                                ▽
 ◁|                                                          |▷
```
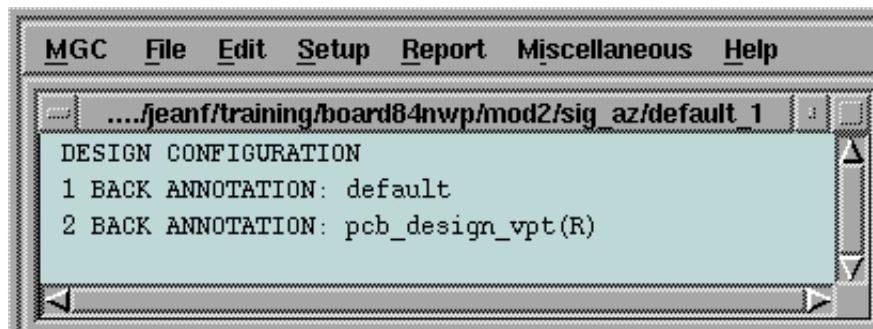
Figure 3-40. Two Back Annotations Connected

6. Click on the **Palette > Save Design** icon.

The engineering viewpoint is saved with the back annotation objects connected.

7. Click on the **Palette > Close Vpt** icon.

This closes all design windows, but leaves DVE open.

8. Close the DVE Session window and the Design Manager windows.

You have now completed this module. Continue with *Board Station for New Users Training Series*, "Module 3: Creating PCB Geometries".

欢 迎 光 临

http://www.mweda.com   http://edastudy.ik8.com

本站专业提供各种微波仿真软件和 PCB 设计软件的安装光盘和学习培训教程，本站提供的所有教程教材都是公司培训用的，很系统的讲述了相关软件的应用。主要项目有：

破解软件类：微波仿真软件

ADS2004A   ADS2003C   ADS2003A

HFSS9.2   HFSS9.1   HFss9.0   HFSS8.0

Ansoft Designer1.1   Ansoft Serenade8.71   Ansoft Maxwell 10   Ansoft SIWave

Microwave Office2002   Sonnet Suite Pro 9.52   CST5.0

Super NEC 2.5   Zeland IE3D9.2   XFDTD 6.0

破解软件类：PCB 工具软件

Mentor EN2004   Mentor EN2002   Mentor ePD2004   Mentor SDD2004

Mentor WG2004   Mentor WG2002   Mentor ISD2004

PowerPCB5.0   PowerLogic5.0   Orcad10.3

PADS2005   PADS2004

Cadence SPB15.2 (Allegro 15.2)   Cadence PSD15.0 (Allegro 15.0)

软件学习、培训教程：

Mentor EN: Mentor EN 原版培训教程

Mentor EN 视频教程

Mentor WG: Mentor Expedition / Mentor WG 中文用户手册

Mentor Expedition（WG）PCB Training Workbook

Mentor DxDesigner Design Processing Training Workbook

HFSS：   HFSS 9.2 入门与提高教程

HFSS 9.0 入门与提高教程

HFSS9 视频教程

HFSS9 Training Tutorials

HFSS8 Training Manual

电子科大 HFSS8 教程

ADS: ADS 中文基础教程

ADS 设计实验教程

ADS 入门教程

ADS2003 Training Workbook: ADS2003 fundamental

ADS Training Workbook for Momentum

ADS Customization Training Workbook

注： 本站还有Ansoft Maxwell, Ansoft Serenade, Ansoft Q3D，Zeland IE3D, PowerPCB，
Orcad等的培训教程。详情可登陆：http://www.mweda.com 或 http://edastudy.ik8.com

欢 迎 光 临

　　本站专业提供各种微波仿真软件和 PCB 设计软件的安装光盘和学习培训教程，本站提供的所有教程教材都是公司培训用的，很系统的讲述了相关软件的应用。主要项目有：

破解软件类：微波仿真软件

ADS2004A　　　ADS2003C　　　ADS2003A

HFSS9.2　　　　HFSS9.1　　　　HFss9.0　　　　HFSS8.0

Ansoft Designer1.1　　　Ansoft Serenade8.71　　　Ansoft Maxwell 10　　　Ansoft SIWave

Microwave Office2002　　　Sonnet Suite Pro 9.52　　　CST5.0

Super NEC 2.5　　　　Zeland IE3D9.2　　　　XFDTD 6.0

破解软件类：PCB 工具软件

Mentor EN2004　　　Mentor EN2002　　　Mentor ePD2004　　　Mentor SDD2004

Mentor WG2004　　　Mentor WG2002　　　Mentor ISD2004

PowerPCB5.0　　　PowerLogic5.0　　　Orcad10.3

PADS2005　　　　PADS2004

Cadence SPB15.2 (Allegro 15.2)　　　Cadence PSD15.0 (Allegro 15.0)

软件学习、培训教程：

Mentor EN:　　Mentor EN 原版培训教程

　　　　　　　　Mentor EN 视频教程

Mentor WG:　　Mentor Expedition / Mentor WG 中文用户手册

　　　　　　　　Mentor Expedition（WG）PCB Training Workbook

　　　　　　　　Mentor DxDesigner Design Processing Training Workbook

HFSS：　　　HFSS 9.2 入门与提高教程

　　　　　　　HFSS 9.0 入门与提高教程

　　　　　　　HFSS9 视频教程

　　　　　　　HFSS9 Training Tutorials

　　　　　　　HFSS8 Training Manual

　　　　　　　电子科大 HFSS8 教程

ADS:　　　　ADS 中文基础教程

　　　　　　　ADS 设计实验教程

　　　　　　　ADS 入门教程

　　　　　　　ADS2003 Training Workbook: ADS2003 fundamental

　　　　　　　ADS Training Workbook for Momentum

　　　　　　　ADS Customization Training Workbook

注： 本站还有Ansoft Maxwell, Ansoft Serenade, Ansoft Q3D，Zeland IE3D, PowerPCB，
Orcad等的培训教程。以及射频/微波/天线类英文原版书籍。详情可登陆：
http://www.mweda.com 或 http://edastudy.ik8.com